000475

THE
PADS™-LOGIC
USER MANUAL

CAD SOFTWARE, INC.
119 Russell Street
Suite #6
LITTLETON, MA 01460

REV 0 1/15/90

CAD Software, Inc. Copyright (C) 1990
-- All rights reserved.

This program is the property of and embodies the proprietary trade secrets of CAD Software, Inc. Unauthorized use and copying are prohibited by law including United States and foreign copyright law.

Credits:

*IBM is a registered trademark of IBM Corp.
FutureNet is a registered trademark of DATA I/O.

PADS-LOGIC
USER MANUAL

TABLE OF CONTENTS

PART I

THE INTRODUCTION

PART II

THE SELF-TRAINING TUTORIALS

PART III

THE REFERENCE MANUAL

PADS-LOGIC
USER MANUAL

TABLE OF CONTENTS

PART I - INTRODUCTORY SECTION

CHAPTER 1 - GETTING STARTED

1.0 INTRODUCTION 1
1.1 THE ENGINEER'S INTERFACE AT THE CRT 1
1.2 USER MANUAL ORGANIZATION 2
1.3 UNIQUE FEATURES OF ***PADS-LOGIC*** 3
1.4 USER MANUAL ORGANIZATION 5
1.5 CAE DIRECTORY PATH 6
1.6 TRAINING FILES 8
1.7 HARDWARE 8
1.8 LOADING ***PADS-LOGIC*** SOFTWARE 8
1.9 WARRANTY AND SUPPORT 9

PADS-LOGIC
USER MANUAL

TABLE OF CONTENTS

PART II - SELF-TRAINING TUTORIALS

INTRODUCTION

TUTORIAL ONE:
GETTING ACQUAINTED WITH ***PADS-LOGIC***
1.0 INTRODUCTION 1
1.2 INITIAL CRT PRESENTATION 1
1.3 CRT FEATURES 4
1.4 COMMAND MENU WINDOW 5
1.5 COMMAND MENU SELECTION 6
1.6 SYSTEM GRID VS. DOT GRID 7
1.7 CRT MANIPULATION 8
1.7.1 REPAINT 9
1.7.2 RESET 9
1.7.3 PAN 9
1.7.4 AUTO PAN 10
1.7.5 EXPAND 10
1.7.6 CONTRACT 10
1.7.7 CENTERING 10
1.7.8 STOP CRT REPAINT 11
1.7.9 CREATING A BOX 11
1.7.10 COMMAND MENU MACROS 11
1.8 PRACTICE ASSIGNMENT 11
1.8.1 PRACTICE: SETTING UP A NEW SCHEMATIC 12
1.8.2 PRACTICE: SAVING YOUR WORK 13

TUTORIAL TWO:
PLACING PARTS ON THE SCHEMATIC
2.0 INTRODUCTION 1
2.1 ADDING PARTS 4
2.2 ALTERNATE 8
2.3 COPY PART 8
2.4 MOVE PART 9
2.5 MOVE PART TEXT 10

2.6 DELETE 11
2.7 PART TEXT EDITING 11
2.8 REFERENCE DESIGNATIONS 11
2.9 PIN (PAD) NUMBERS 12
2.10 PART TYPE 13
2.11 SET UP PART 15
2.12 SEARCH MACRO 16
2.13 PRACTICE: PLOTTING YOUR WORK 16
2.14 PRACTICE: SAVING YOUR WORK 16

TUTORIAL THREE:
ADDING CONNECTIONS TO THE SCHEMATIC
3.0 INTRODUCTION 1
3.1 ADD CONNECTION 1
3.2 PRACTICE: STARTING A CONDUCTOR FROM A TERMINAL 3
3.3 PRACTICE: FROM A CONDUCTOR 3
3.4 USING OFF-PAGE TAG 4
3.5 MOVE CONNECTIONS 5
3.6 EDIT TEXT (RENAME CONNECTIONS) 6
3.7 DELETE NET 7
3.8 SETUP CONNECTIONS 8

TUTORIAL FOUR:
ADDING BUSES TO THE SCHEMATIC
4.0 INTRODUCTION 1
4.1 MOVE BUS 2
4.2 ADDING CONNECTIONS TO A BUS 2
4.3 GROUP BUS 4
4.4 DELETE BUS 4

TUTORIAL FIVE:
ADDING 2-D LINES AND TEXT
5.1 2-D LINES 1
5.1.1 NEW POLY 2
5.1.2 CREATE NEW CIRCLE 2

5.1.3 MODIFY LINE 2
5.1.4 COPY LINES 3
5.1.5 MOVE LINE 3
5.1.6 DELETE LINE 4
5.1.7 COMBINE LINES AND TEXT 4
5.1.8 FROM LIB (LINES) 5
5.1.9 TO LIB (LINES) 6
5.2 ADD TEXT 6
5.2.1 COPY TEXT 8
5.2.2 EDIT TEXT 9
5.2.3 DELETE TEXT 9
5.2.4 MOVE TEXT 9
5.2.5 COMBINE TEXT AND 2-D LINES 10

TUTORIAL SIX:
USING THE GROUP AND PLOT FEATURES
6.1 GROUP 1
6.1.1 MOVE GROUP 2
6.1.2 COPY GROUP 2
6.1.3 GROUPING OF HIERARCHICAL SYMBOLS 3
6.1.4 DELETE GROUP 3
6.1.5 GROUP DELETE WITH
HIERARCHICAL SYMBOLS 4
6.1.6 ADD AND DELETE 4
6.1.7 CUT GROUP 5
6.1.8 GROUP BUS 6
6.2 PLOTTING YOUR SCHEMATIC 7

TUTORIAL SEVEN:
MACRO ROUTINE
7.1 INTRODUCTION 1
7.2 MACRO COMMANDS 1
7.3 TYPES OF MACROS 2
7.4 CREATING MACROS 2
7.5 CREATING AN AUTO MACRO 3
7.6 LOADING USER MACROS 4
7.7 CONTROLLING MACRO REPLAY 4

7.8 USER MACROS 4
7.8.1 WHAT ARE MENU MACROS 5
7.8.2 MACRO LANGUAGE 6
7.8.3 USER DESIGN MACROS 7

TUTORIAL EIGHT:
USING THE HIERARCHICAL ROUTINE.

TUTORIAL NINE:
USING THE THE LIBRARY ROUTINE
9.1 CREATION OF A MULTI-GATED
(74LS02 IC) COMPONENT 1
9.2 CREATION OF A DISCRETE PART 7
9.3 CONNECTOR 12
9.4 CREATING A NEW GATE DECAL 16
9.5 THE PKGTYP.DAT FILE 18
9.6 DELETING A PART 19

TUTORIAL TEN:
FILES MANAGEMENT
10.1 OBJECTIVE 1
10.2 JOB DIRECTORY LISTINGS 1
10.3 JOB CONTROL PROCEDURE 2
10.4 DELETION OF FILES 3
10.5 DAILY BACKUP 4
10.5.1 MAKING A BACKUP FLOPPY 4
10.5.2 FILES ON FLOPPY DISK 6
10.5.3 USING THE FILES DISK 6
10.6 ARCHIVAL FILES 6
10.7 STORAGE SPACE 8

PADS-LOGIC
USER MANUAL

PART III - REFERENCE MANUAL

CHAPTER 1 - CRT PRESENTATION

1.1 PURPOSE 1
1.2 PADS-LOGIC PROGRAM INITIALIZATION 1
1.3 ***PADS-LOGIC*** CRT PRESENTATION 3
1.4 CRT FEATURES 4
1.4.1 CURSOR 5
1.4.2 WORKING AREA 5
1.4.3 CURSOR POSITION INDICATOR WINDOW 6
1.4.4 DRAWING NAME, SHEET AND GRID DATA 6
1.4.5 ACTIVITY WINDOW 7
1.4.6 COMMAND MENU TITLE BLOCK 8
1.4.7 COMMAND MENU WINDOW 8
1.4.8 MENU MACROS 12
1.4.9 SYSTEM PROMPT WINDOW 13
1.5 CRT MANIPULATION 13

CHAPTER 2 - IN/OUT ROUTINE

2.0 INTRODUCTION 1
2.1 BINARY JOB VS. ASCII FORMATS 1
2.2 JOB IN SUBROUTINE 2
2.3 JOB OUT SUBROUTINE 3
2.3.1 BACKUP OR SECURITY FILE 4
2.3.2 NOTES ON .SCH FILES 6
2.4 ASCII IN SUBROUTINE 7
2.5 ASCII OUT SUBROUTINE 8
2.6 SHEETS 10
2.6.1 SELECT SHEET 10
2.6.2 ADD SHEET 11
2.6.3 RENUMBER SHEET 12
2.6.4 DELETE SHEET 12
2.7 ENGINEERING CHANGE ORDER ROUTINE (ECO) 13
2.7.1 TO PCB 14
2.7.2 FROM PCB 15
2.8 REPORTS 16
2.8.1 NET LIST 17
2.8.2 BILL OF MATERIAL 22

2.8.3 OFF-PAGE LIST 26
2.8.4 PART/NET STATUS 28
2.8.5 UNUSED REPORT 29
2.8.6 JOB LIMITS REPORT 31
2.9 PLOT ROUTINE 33
2.9.1 USING THE PLOT ROUTINE 33
2.9.2 PHOTOPLOTTER 37
2.9.3 APERTURE 37
2.10 NEW SCHEMATIC 39

CHAPTER 3 - ADD PART ROUTINE
3.1 ADD PART 1
3.2 SUBROUTINES OF ADD PART 2
3.2.1 COMPLETE 2
3.2.2 ROTATE 2
3.2.3 MIRROR X 2
3.2.4 MIRROR Y 3
3.2.5 ALTERNATE 3
3.2.6 NEXT TYPE 3
3.3 NOTES ON USING ADD PART 3

CHAPTER 4 - ADD CONNECTION ROUTINE
4.1 CRN/FINISH 2
4.2 DELETE CORNER 2
4.3 ANGLE 2
4.4 OFF PAGE 3
4.5 GROUND 3
4.6 POWER 4

CHAPTER 5 - THE EDIT TEXT ROUTINE
5.0 INTRODUCTION 1
5.1 EDIT TEXT 1
5.1.1 REF DES 2
5.1.2 PART TYPE 2
5.1.3 PIN SWAP 3
5.1.4 PART TOLERANCE/VALUES 4
5.2 FREE TEXT 4

5.3 NET NAMES 4
5.4 BUSES 5

CHAPTER 6 - COPY ROUTINE
6.0 INTRODUCTION 1
6.1 PART 1
6.2 NOTES FOR USING COPY 2
6.3 TEXT 2
6.4 2-D LINES 2
6.5 HIERARCHICAL SYMBOLS 2

CHAPTER 7 - MOVE AND DELETE ROUTINES
7.0 INTRODUCTION 1
7.1 MOVE 1
7.1.1 MOVE PART 1
7.1.2 MOVE CONNECTION 2
7.1.2.1 MOVE SEGMENT 2
7.1.2.2 MOVE CORNER 3
7.1.2.3 DISCONNECT PIN 4
7.1.3 MOVE BUS 4
7.2 DELETE 4
7.2.1 NOTES ON USING THE DELETE ROUTINE 5

CHAPTER 8 - SETUP ROUTINE
8.1 INTRODUCTION 1
8.2 CRT SETUP SELECT 2
8.3 PARAMETERS 4
8.3.1 DOT GRID 5
8.3.2 REAL WIDTH 6
8.3.3 BACK UP INTERVAL 6
8.3.4 SHEET SIZE 7
8.3.5 PAN 8
8.3.5.1 INTRODUCTION 8
8.3.5.2 PAN ENABLE Y/N 8
8.3.5.3 PAN SENSITIVITY 8
8.3.5.4 PAN DELAY 8

8.3.5.5 PAN DISTANCE 9
8.3.6 BUS LINE WIDTH 9
8.3.7 BUS ANGLE OFFSET 9
8.3.8 CONNECTION WIDTH 9
8.3.9 TIE DOT DIAMETER 9
8.3.10 NET NAMES-HEIGHT, LINE WIDTH 10
8.3.11 PIN NUMBER 10
8.3.12 PIN NAMES 10
8.3.13 2-D ITEMS 10
8.4 MACROS 10

CHAPTER 9 - ADD BUS ROUTINE
9.0 INTRODUCTION 1
9.1 ADD BUS 1
9.1.1 NOTES ON BUSES 3

CHAPTER 10 - ADD TEXT ROUTINE

CHAPTER 11 - 2-D LINES ROUTINE
11.0 INTRODUCTION 1
11.1 NEW POLY 1
11.2 NEW CIRCLE 2
11.3 MODIFY 3
11.4 COMBINE 3
11.5 COPY 4
11.6 MOVE 4
11.7 DELETE 4
11.8 FROM LIB 4
11.9 TO LIB 5

CHAPTER 12 - GROUP ROUTINE
12.0 INTRODUCTION 1
12.1 DEFINE GROUP 1
12.2 PASTE 5
12.3 GROUPING OF HIERARCHICAL SYMBOLS 5

12.4 GROUP DELETE WITH HIERARCHICAL SYMBOLS 6

CHAPTER 13 - THE PARTS LIBRARY MANAGER (PLIB)
13.0 INTRODUCTION 1
13.1 ANATOMY OF A ***PADS-LOGIC*** PART 1
13.1.1 ELECTRICAL DATA 2
13.1.2 GATE INFORMATION 4
13.1.3 SCHEMATIC (CAE) DECAL 5
13.1.4 PCB DECAL 5
13.2 PLIB 5
13.3 2-D LINES DATABASE 7
13.4.1 PCB DECALS DATABASE 7
13.4.2 CAE DECALS DATABASE 8
13.5 PART TYPE DATABASE 8
13.6 USE OF THE PLIB EDITOR 8
13.7 P (PART TYPE LIBRARY) 9
13.8 INITIALIZE 13
13.9 SYSTEM PARTS LIBRARY 13
13.10 MAINTENANCE OF SYSTEM PARTS LIBRARY 14
13.10.1 CREATING OR MODIFYING PART DECALS/TYPES 14
13.10.2 BACKUP COPIES 14
13.10.2.1 USER LIBRARY 15
13.10.2.2 STANDARD LIBRARY 16
13.11 SUMMARY OF ***PADS-LOGIC*** LIBRARY ORGANIZATION 16

CHAPTER 14 - THE PARTS LIBRARY
14.0 INTRODUCTION 1
14.1 PART TYPE 3
14.1.1 ELECTRICAL 4
14.1.2 MOD GATE 10
14.1.2.1 2-D LINES 12
14.1.2.2 ADD TEXT 13
14.1.2.3 TERMINAL 13
14.1.2.3.1 ADD TERMINAL 13

14.1.2.3.2 CHG DECAL 14
14.1.2.3.3 CHG SEQ NO 14
14.1.2.3.4 SET PNUM 14
14.1.2.3.5 SET PNAME 15
14.1.2.3.6 SET PTYPE 15
14.1.2.3.7 SET PSWAP 16
14.1.2.3.8 CHG PNUM 16
14.1.2.3.9 CHG PNAME 17
14.1.2.4 SETUP 17
14.1.2.5 COPY 17
14.1.2.6 MOVE 18
14.1.2.7 DELETE 18
14.1.2.8 ORIGIN 18
14.1.2.9 COMPLETE 19
14.1.3 CHECK PART 19
14.1.4 SAVE PART 19
14.1.5 NON-AUTOMATIC CREATION OF A GATE DECAL 20
14.1.6 MODIFYING EXISTING PARTS AND GATES 21
14.2 CONNECTOR 22
14.2.1 ELECTRICAL 23
14.2.2 MOD GATE 24
14.3 OFFPAGE 26
14.3.1 OFFPAGE 27
14.3.2 GROUND 28
14.3.3 POWER 29
14.4 REP OFFPAGE 29
14.5 GATE DECAL 29
14.6 PIN DECAL 32
14.7 THE PKGTYP.DAT FILE 34

CHAPTER 15 - THE HIERARCHICAL ROUTINE
15.0 HIERARCHICAL BACKGROUND 1
15.1 NEW HIERARCHICAL COMPONENT 4
15.2 MODIFY HIERARCHICAL COMPONENT 12
15.3 ADD SHEET NUMBER 13
15.4 CREATING THE CIRCUIT CONTENTS OF A HIERARCHICAL SYMBOL 13

15.4.1 HIERARCHY IN 13
15.4.2 HIER OUT 14
15.5 NOTES ON HIERARCHICAL LEVELS 14
15.6 FLATTENED DESIGN 15
15.7 SUB-SECTIONS OF A HIERARCHICAL DESIGN 15

CHAPTER 16 - THE MACRO ROUTINE
16.1 WHAT IS A MACRO? 1
16.2 CREATING A MACRO 2
16.3 DISPLAY RESOLUTION INDEPENDENCE AND COORDINATES WITHIN MACROS 2
16.4 ABSOLUTE VERSUS RELATIVE COORDINATES 2
16.5 THE AUTO MACRO 3
16.6 LOADING A MACRO FILE 3
16.7 MACRO REPLAY CONTROL 3
16.8 YES/NO QUESTIONS 3
16.9 YES/NO RESTORE MODE 4
16.10 MACROS CAN CALL OTHER MACROS 5
16.11 MACRO COMMANDS 5
16.12 CREATE AUTO/USER MACRO 5
16.13 SET ABSOLUTE/RELATIVE COORDINATE MODE 6
16.14 RESTORE MENU POSITION 6
16.15 TOP MENU 6
16.16 DISPLAY FILE 6
16.17 DISPLAY ERRORS 7
16.18 SET MACRO REPLAY COUNT 7
16.19 HELP/STATUS 7
16.20 LOAD FILE 8
16.21 SAVE FILE 8
16.22 DELETE A MACRO 8
16.23 DEFAULTS 8
16.24 LIMITS 8
16.25 MACRO LANGUAGE 9
16.26 SAMPLE MACROS 11
16.27 MODELESS COMMANDS 12

PADS-LOGIC
USER MANUAL

APPENDIX A - LIBRARY PARTS

APPENDIX B - DECAL PLOTS

APPENDIX C - NETWORKING

PART I

PADS™-LOGIC

INTRODUCTION

PART I

CHAPTER 1

GETTING STARTED

1.0 INTRODUCTION

CONGRATULATIONS!! You have purchased ***PADS-LOGIC***, an advanced Logic Capture and Schematic CAE package that has a number of features that are unique to the industry. Now that you have ***PADS-LOGIC*** your next task is to learn how to use it effectively--and to do this in the least possible time. These objectives are the purpose of this User Manual.

We at CAD Software, Inc., have made a major effort in developing a self-training approach that will enable you to learn to use the system without great expenditure of time and effort on your part. The approach we use is a dual effort involving (1) a highly efficient designer interface at the CRT, and (2) special attention to the organizational contents of this User Manual.

The important thing for both yourself and CAD Software, Inc. is that you read and study this manual, especially Part II, The Tutorial Section, BEFORE rushing into trying to create your first schematic. Experience with over several thousand users of the *PADS* products has shown that the single most common problem is the User that attempts to use the sophisticated software without adequate study. Indeed, we have Hot-line support at our Littleton, MA, facility that is available to you for 30 days after you purchase the software and while you are learning the system. But beware! Don't use this Hot-line support service as a substitute for using this manual and following the self-training exercises contained herein. Please! This User Manual has been prepared for your use. Use it, please.

1.1 THE ENGINEER'S INTERFACE AT THE CRT

The Interface between the Engineer and the CRT is self-coaching and easy to use. The system commands are self-evident and familiar

to an Engineer.

To simplify the Engineer's Interface, the Command Menu Structure has been organized into routines and subroutines which duplicate the engineer's thought process while converting the design to paper, i.e., selection of a part, placing the parts in a logical sequence on the schematic, connecting up the parts, etc. All the Menu commands associated with a routine have been grouped together so that at any one time only those commands pertinent to the activity underway are presented to the Engineer for decision and action as necessary.

A unique feature to the ***PADS-LOGIC*** Engineer's Interface is a "Message Window" at the bottom of the screen which continually coaches the Engineer as to the information required for the particular action, including the "format" of the Engineer's response.

Finally, the elapsed time to implement Interactive commands is very fast--thus the user does not become frustrated waiting for the CPU response. In short, the Engineer's Interface is extremely user friendly, fast, and easy to use--you will enjoy using it!

1.2 USER MANUAL ORGANIZATION

This User Manual is arranged into three parts:

PART I, Introductory Section

This Introductory Section orients you as to the overall concept of the ***PADS-LOGIC*** Self-Training Manual, the software and ongoing help from CAD Software, Inc.. A number of subjects are covered: (1) The Unique Features of ***PADS-LOGIC***, (2) The Organization of this Manual, (3) Directories, (4) Hardware Requirements; and finally, (5) A discussion is presented regarding ongoing Support for ***PADS-LOGIC***.

PART II, Tutorial Section

The Tutorial Section, containing Tutorial or "Do-It-Yourself" instructions on every step in capturing the Logic of your design, leads you through the Self-Training process of creating a schematic. We have included a multi-sheet practice Schematic that, while it does

not represent an actual design, nonetheless, includes virtually every situation an Engineer would encounter during the capturing of his design. As a result, upon completion of the training schematic, you will have a keen appreciation of the capability of the ***PADS-LOGIC*** software in virtually any situation encountered during the creation of a Logic Diagram or Schematic.

PART III, Reference Manual

This section of the User Manual is a Reference Manual. Each routine and subroutine is described in complete detail in this Reference Manual. There are a number of Sections in Part III, with each Section dedicated to a specific routine and then subdivided into the subroutines associated with the major routine. If at any time during your self-training or subsequent use, you have a question, simply refer to the Reference Manual (PART III) and the Section pertaining to the main routine, describing the topic to which your question pertains.

1.3 UNIQUE FEATURES OF *PADS-LOGIC*

The ***PADS-LOGIC*** software contains a number of features that are unique not only to ***PADS-LOGIC*** but the CAE industry in general.

First, the database used by ***PADS-LOGIC*** has simultaneous access to all sheets of the design in Memory on a sheet-by-sheet basis, without accessing the hard disk. We call this our "Design-Oriented Database" concept. Most other CAE systems store the contents of only a single sheet in memory at any one time. This is a severe restriction; ***PADS-LOGIC*** does NOT have this handicap. With our "Design-Oriented Database," ***PADS-LOGIC*** has immediate access to the entire design, including all sheets. The advantages of a "Design-Oriented Database" over a "sheet-oriented system" include the following:

- The Engineer can immediately display ANY sheet of the drawing at the CRT without the long, time consuming effort involved in bringing a new sheet in from disk.

- The design-oriented database provides for Automatic

assignment of Reference Designations. You do not need to type in Reference Designations or keep track of what Reference Designations have been used or not been used, etc.

- The Design-Oriented Database keeps track of the gates used in multi-gate Integrated Circuits, such that you never duplicate the use of a gate. Gates are always assigned correctly to the correct Part. Moreover, the system assigns the correct pin number to the Part Pin.

- Batching of sheets at the completion of the design to generate Net lists and Part lists is NOT required with the ***PADS-LOGIC*** design-oriented database. Instead, the complete Net list and Part list across the entire design is available instantly, at any time.

A second and significant feature unique to ***PADS-LOGIC*** is the concept of Forward and Backward annotation between the Logic Capture system and the Printed Circuit Board CAD system, including Engineering Change Orders (ECO).

Upon completion of the Logic Capture effort, a file containing both the Parts and the Connections is immediately available to "forward annotate" the data to a Printed Circuit Board Layout CAD system, Logic Simulation & Circuit Analysis. Powerful Macro commands and a multi-level Hierarchical Level are available for the sophisticated user.

An Engineering Change Order (ECO) feature is available which collects or batches together the various changes the Engineer may make to a Schematic into an ECO batch file that can be sent forward to the Printed Circuit Board CAD system for the updating of the board database originating during the original forward annotation data from ***PADS- LOGIC***.

Similarly, changes made in the board CAD system in the form of Gate Swapping, Pin Swapping, and changes to the Reference Designations, can be used by the ***PADS-LOGIC*** software to automatically update the original schematic, created in ***PADS-LOGIC***.

To summarize: ***PADS-LOGIC*** has a number of unique features that set it aside from most of today's Logic Capture systems. We think you will enjoy using these features!!

1.4 USER MANUAL ORGANIZATION

A word now about the organization of the contents of this ***PADS-LOGIC*** User Manual.

PART II TUTORIALS contain the following Tutorials:

Tutorial One: Getting Acquainted

Provides detailed information as to the CRT presentation, how to manipulate the data at the CRT, and the Command Menu Structure.

Tutorial Two: Part Symbol Placement

Provides the engineer with a Self-Training Exercise in the placement of Parts associated with the training schematic provided by CAD Software.

Tutorial Three: Connections

Provides training on all aspects of hooking up the connections between parts.

Tutorial Four: Buses

Provides training on adding Buses to the Schematic.

Tutorial Five: 2-D Lines and Text

Provides training on adding 2-D Lines and Free Text to the Schematic.

Tutorial Six: Group and Plotting

Provides training in the "Group" feature and plotting of the Schematic.

Tutorial Seven: Macro Commands

Provides training in using the Macro facility.

Tutorial Eight: Hierarchy

Provides training in using the Hierarchical feature.

Tutorial Nine: **PADS-LOGIC** Library

Provides training in using the **PADS-LOGIC** Parts Editor.

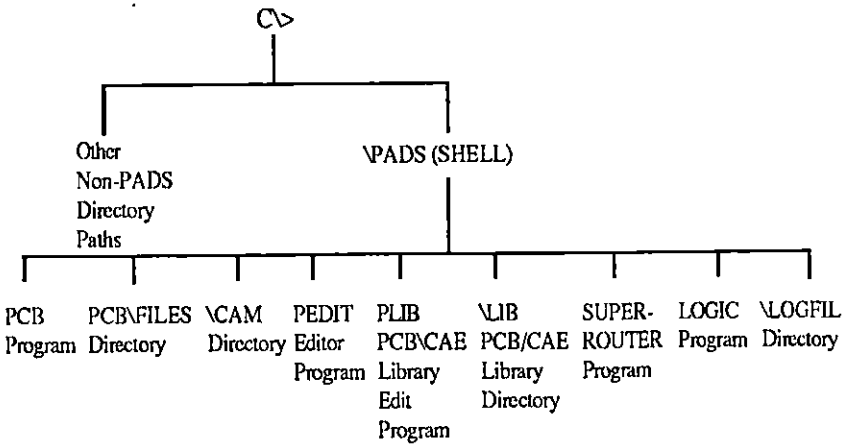
Tutorial Ten: Files Management

Provides training in managing the files created.

1.5 CAE DIRECTORY PATH

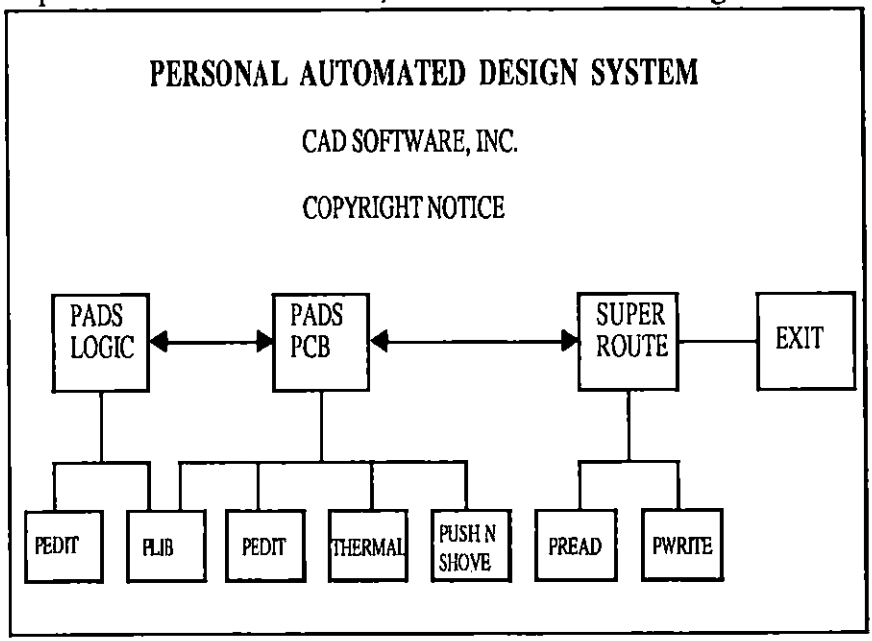
All the programs associated with ***PADS-LOGIC*** are contained within its own Directory Path (if you are a new DOS user, you should familiarize yourself with the DOS concept of Directory Paths, Directories and SubDirectories.).

In the following, the figure illustrates the programs and subdirectories associated with ***PADS-LOGIC*** .



DIRECTORY PATH

PADS-PCB and ***PADS-LOGIC*** can reside on the hard disk with other programs. To enter the PADS Shell from the DOS root directory (\), or from other directories, type PADS <CR>. The response is the PADS SHELL, as shown in the following:



Within the PADS SHELL directory, there are programs and their associated options and directories. These are (1) Logic Capture program (***PADS-LOGIC***), (2) the PCB Design Program (PCB), and (3) the Super Router program.

To enter the LOGIC program, be in the PADS SHELL and, using the Mouse Cursor, select the ***PADS-LOGIC*** box. The ***PADS-LOGIC*** Copyright notice and "Press any Key" will appear. Press any key and the ***PADS-LOGIC*** top menu will be displayed at the CRT.

All Logic Capture files created in the ***PADS-LOGIC*** program are stored in a subdirectory called \PADS\LOGFIL. To access this directory, enter CD \PADS\LOGFIL.

LOGIC has two utilities: (1) PEDIT, a special purpose ASCII Editor, and (2) PLIB, the Library Maintenance Program.

PADS-PCB and **SUPERROUTER** are Board Layout and Auto Routing programs described in their respective manuals.

1.6 TRAINING FILES

The following files will be used during your self-training and have been included in the sub-directory PADS\LOGFIL:

DEMO.SCH
SELFTRAN.SCH

1.7 HARDWARE

1. An IBM-AT 286, a 386 or true compatible as approved by CAD Software, Inc.
2. 640K of Expanded Memory
3. 640K DOS Memory
4. COM1: and COM2: Ports
5. CAD Software, Inc. approved EGA or VGA Graphics Board
6. A 3-button Mouse (NOTE: A 2-button mouse may be used, however, is not recommended. Contact CAD Software, Inc. Technical Support for instruction on how to set up the 2-button mouse.)
7. An EGA or VGA Color Monitor
8. CAD Software approved Matrix Printer

DO NOT USE a Dual CRT configuration or a configuration not approved by CAD Software, Inc.

NOTE: If ***PADS-LOGIC*** is used with **PADS-PCB**, the version of **PADS-PCB** must be 3.2, or higher.

NOTE: PADS-LOGIC is a general purpose Logic Capture program and can be used with most Board CAD systems.

1.8 LOADING PADS-LOGIC SOFTWARE AND GETTING ORGANIZED

At this time you should have your computer loaded with the ***PADS-LOGIC*** software using the installation program PINSTALL, and the PADS Installation Manual; the computer

should be operational and you should be ready to start your self-training.

Each of the PART II Self-training Tutorials are structured for a particular activity. We suggest that you first read this Part I INTRODUCTION, to obtain an overview; then spend your time on each of the Tutorials in PART II, Self-Training Tutorials.

1.9 WARRANTY AND SUPPORT

At the end of this chapter is a gray card and a yellow card. The gray card is your 30-day Warranty Registration card. As a part of your software purchase, you are entitled to assistance during the initial loading of your software and for 30 days subsequent to the purchase while you are using the software. This is available only if you send in the gray 30-day Warranty Registration card. NOW, while you are thinking about it, we suggest you mail back the gray 30-day Warranty Registration card to CAD Software, Inc..

The yellow card describes an Extended Support Contract which CAD Software offers you, to go into effect upon the completion of your 30-day Warranty Registration period. This Extended Support Contract is for the following 12 months and provides unlimited telephone consulting services with our Hot-line Support Group for any problems or questions you may have. You will receive no-cost updates of all enhancements or improvements as well as bug fixes that are released during the next year. As a suggestion, before your 30-day warranty period is over, give considerable thought to the purchase of the Extended Support Contract. You will find it most worthwhile.

000475

Your Registration Serial Number

PADS-LOGIC EXTENDED SUPPORT CONTRACT

As a user of ***PADS-LOGIC*** you may, upon completion of the 30-day warranty registration, purchase an Extended Support contract for the following 12 months. The cost of this contract is $150.00 per year (U.S. and Canada) and the benefits far outweigh this small cost. The income from this Extended Support is used for:

(1) Maintaining a staff of specialists to respond to any questions you may have, and

(2) NO-COST enhancements to your ***PADS-LOGIC*** Software.

A ***PADS-LOGIC*** Extended Support Contract entitles you to the following:

(1) Unlimited telephone consulting on any problems or questions you may have relative to potential software bugs, special work-arounds you may want to use for unique or new applications, etc.

(2) Written correspondence concerning any questions or problems you have regarding the use of the software.

(3) Free updates of the ***PADS-LOGIC*** software you have purchased. One or more updates per year of the software are issued, incorporating enhancements or improvements as well as fixes for bugs reported by yourself and other users throughout the world.

The ***PADS-LOGIC*** Extended Support Contract is applicable to ***PADS-LOGIC*** . It is not applicable to the ***PADS-PCB*** or the ***PADS-SUPERROUTER*** software.

How to Order: In order to maintain uninterrupted continuity with your 30-day Support Agreement, use the following procedure:

(1) Complete the Extended Support Contract today and mail to CAD Software, Inc., 119 Russell Street, Suite #6, Littleton, MA 01460 with your check for $150.00 (or use Visa/ MasterCard or American Express).

(2) CAD Software, Inc. will start your Extended Software Contract and enter your account into our Support computer.

(3) At any time you need to contact us, call our Hot Line (508) 486-3328 and provide us with your Registration Serial Number. We will assist you with whatever questions you may have.

(4) If you have purchased your software and support from a CAD Software, Incorporated authorized Dealer, you should seek help from your local dealer. If your local dealer is not in a position to resolve your problem, please contact us at (508) 486-3328.

(5) We maintain a modem over which you can send us Files you wish us to investigate. The Modem Number is (508)486-9549. Call before sending a file.

PADS-LOGIC
EXTENDED SUPPORT CONTRACT

000475

MAIL TO: CAD Software, Inc.
119 Russell Street
Suite #6
Littleton, MA 01460

or FAX TO:
(508)486-8217

_____
Your Registration Serial Number

1. Enroll me in an Extended Support Contract for my ***PADS-LOGIC*** software. Payment of $150.00 is enclosed___. Charge my American Express/Visa/Master Charge Account No._____,
Expiration date_____.

2. My hardware is:

	Vendor	Model
CPU	_____	_____
Memory	_____	_____
Mouse	_____	_____
Printer	_____	_____
Graphics Card	_____	_____
Printer	_____	_____

3. My software purchased _____
(date)

RETURN TO:

_____
Name

_____
Company

_____
Address

_____

_____

Telephone No._____Ext.____
Fax No._____

CHANGE OF ADDRESS

We at CAD Software, publish a quarterly Newsletter, send out new software updates, prepare letters on new products and new releases, etc.

These items are sent directly to the original person to whom the software was sent when purchased.

If that person is promoted out of CAD, or changes department or corporation, this very informative and valuable information sent out by CAD Software may very well become lost. We ask you, therefore, if there is a change in the "responsible" person for the PADS system, or a change in address, to advise us on this form. In this fashion, we can continue to keep you and your firm advised about new programs, updates, etc.

Please complete the following and return to:

CAD Software, Inc.
119 Russell Street, Suite #6
Littleton, MA 01460

PADS-LOGIC Y/N Serial No._____
PADS-PCB Y/N Serial No._____

Original Responsible Person:
_____Tele:( )_____

Address_____

_____

New Responsible Person:
_____Tele:( )_____

Address_____

_____

PART II

PADS™-LOGIC

SELF-TRAINING TUTORIAL

PART II

INTRODUCTION

SELF-TRAINING TUTORIALS FOR *PADS-LOGIC*

This section of the ***PADS-LOGIC*** Users Manual (Part II) contains a series of Tutorials that you may use in the training of yourself (and others) in the use of ***PADS-LOGIC***.

These Tutorials are organized by Routine or Feature, as used in the creation of a Schematic or Logic Diagram. In the process, they are arranged in a logical step-by-step sequence as one would use the software in the creation of a schematic.

WE URGE YOU TO USE THESE TUTORIALS AS A PART OF YOUR SELF-TRAINING IN THE USE OF ***PADS-LOGIC***. MORE TO THE POINT: DO NOT ATTEMPT TO USE THE SOFTWARE WITHOUT SOME INSTRUCTION AND ORGANIZATION, AS CONTAINED IN THESE TUTORIALS.

From our experience with users of PC-based CAD systems, we can tell you that the most frequently encountered problem with the new user is the rush to the CRT and trying to use the software without some self-training and study of the User Manual.

We hope you will not allow yourself to be caught up with this problem! ***PADS-LOGIC*** is a superb Logic Capture Software; however, it needs your understanding of the program to utilize its best features. So please! Spend some time in your self-training by using "PART II Self-Training Tutorials." We strongly suggest you start with Tutorial One and proceed through all the Tutorials in sequence. As a result, you will have created a multi-sheet Schematic and will have used every feature of ***PADS-LOGIC***.

For your information, PART II contains a number of Self-Training Tutorials. These are listed below:

TUTORIAL ONE:
GETTING ACQUAINTED WITH ***PADS-LOGIC***

TUTORIAL TWO:
PLACING PARTS ON THE SCHEMATIC

TUTORIAL THREE:
ADDING CONNECTIONS TO THE SCHEMATIC

TUTORIAL FOUR:
ADDING BUSES TO THE SCHEMATIC

TUTORIAL FIVE:
ADDING 2-D LINES AND TEXT

TUTORIAL SIX:
USING THE GROUP AND PLOT FEATURES

TUTORIAL SEVEN:
MACRO ROUTINE

TUTORIAL EIGHT:
USING THE HIERARCHICAL ROUTINE

TUTORIAL NINE:
USING THE ***PADS-LOGIC*** LIBRARY

TUTORIAL TEN:
FILES MANAGEMENT

TUTORIAL ONE:

GETTING ACQUAINTED WITH *PADS-LOGIC*

1.0 INTRODUCTION

In this Tutorial, you will be familiarized with the CRT presentation of a sheet of a schematic and the various controls you have for manipulating the presentation, such as Expand, Contract, Pan, Repaint, etc.

NOTE: IN ORDER THAT YOU FOCUS ON THE SUBJECT MATTER OF THIS TUTORIAL, WE STRONGLY RECOMMEND THAT YOU FOLLOW THE INSTRUCTIONS OF THE TUTORIAL EXACTLY AND PRECISELY. DO NOT BE TEMPTED TO "GO OFF ON YOUR OWN," GUESSING WHAT THE VARIOUS MENUS AND FEATURES ARE. BE PATIENT! THEY WILL BE COVERED AT THE APPROPRIATE TIME AND IN THE APPROPRIATE SEQUENCE.

HERE WE GO!!

1.2 INITIAL CRT PRESENTATION

Call the ***PADS-LOGIC*** Application software from the PADS Shell in accordance with "PART 1, INTRODUCTION", Section 1.5, and the menu presentation will be as shown below:

You will note the left side of the CRT, in the menu area, the following:

1 IN/OUT
2 ADD PART
3 ADD CONN
4 EDIT TEXT
5 COPY
6 MOVE
7 DELETE
8 SETUP

9 OTHER MENU
0 EXIT

This is the "Top" or Main Command Menu, and a brief description of each routine is as follows:

IN/OUT:
Used to Recall and Save your schematic files, to set up the sheets of your schematic, to create and accept ASCII files, to generate and accept Engineering Change Orders (ECOs), to generate Reports & Plot your schematic.

IN/OUT is a most important menu; therefore, let's get a little more acquainted at this time. Place the Highlight Bar over IN/OUT by moving the Mouse, then select IN/OUT. Study the various IN/OUT subroutines. Then exit by selecting EXIT, and return to the Top Menu.

ADD PART:
Permits the adding of Parts to a sheet of the schematic.

ADD CONN:
Allows adding of connections to the schematic.

EDIT TEXT:
Allows the editing of any text, including Free Text, Signal Names, and Text associated with Parts.

COPY:
Permits the duplication of Parts, Free Text and Lines.

MOVE:
Permits the manipulating of Parts, Connections, Free Text and Lines.

DELETE:
Permits the deleting of Parts, Text, Connections and Lines, etc., from the database.

SETUP:
Permits the setting up of the Design Environment you use while creating the schematic. Colors used to present items at the CRT are "setup" in this routine.

OTHER MENU:
Extends the Main menu by allowing adding a Bus, adding Free Text, adding Lines, creating a Group, Copying, Moving, Deleting items, Hier/Lib and finally, Setup the design Environment. Note that COPY, MOVE, DELETE and SETUP are identical to the same routines in the Main Menu, and are repeated in OTHER MENU for your functional convenience.

Use the mouse to select each of the above routines, and examine them. Do not use the Routines at this time. Just become familiar with their location. Currently, we are simply becoming acquainted with ALL the routines. Their use will come in later Tutorials.

CALLING UP A SCHEMATIC FROM HARD DISK

You are now going to familiarize yourself with the CRT Presentation and the various commands you have under your control. To do this, we need a Schematic displayed at the CRT.

For this purpose, we have created a schematic called, "DEMO", a multi-page schematic that you will use at this time to familiarize yourself with the CRT presentation and controls. Proceed as follows:

Be in the Main Menu; Select IN/OUT by either selecting the F1 Key or moving the mouse with the highlight bar over the 1 IN/OUT menu and click the Left mouse button. The menu changes. Select F1 JOB IN. In response to the prompt:

JOB INPUT FILE NAME (CR = CAE.SCH)

Respond by typing: DEMO <CR>.

NOTE: <CR> = Carriage Return.

NOTE: If you had simply hit Carriage Return, the Backup or Security file called, CAE.SCH, would have been brought into memory and displayed.

We will talk more about the menu commands later on; but for the time being, all we are interested in is displaying a schematic at the CRT. The schematic called "DEMO" has been created by CAD Software, Inc., and stored as a file for your training. DEMO is shown in Figure 1a, 1b, 1c and 1d (located at the end of this Tutorial).

By the way, after you have entered DEMO at the keyboard, complete your entry with a strike of the Carriage Return key <CR>. This tells the CPU to process your entry.

The Schematic called "DEMO" will now be displayed at the CRT.

1.3 CRT FEATURES

With the schematic displayed at the CRT, let's discuss some of the features at the CRT.

In the upper left-hand corner is a White window which contains the X and Y coordinates of the cursor. Move the cursor, and you will see these coordinates change. The coordinate system is a Cartesian Coordinate System with 0,0 being at the lower left-hand corner of the drawing sheet.

The Purple window in the upper left corner contains three items of information:

1) The name of the file in memory.

2) The sheet number displayed and the number of sheets in the drawing. (2/5 indicates sheet 2 of 5 is currently displayed).

3) The System Grid in 0.001" increments. (Default is Grid = 100)

The Yellow window in the upper left side of the CRT contains a "Postage Stamp" feature. This feature helps you locate yourself

within a large drawing sheet. You will notice the Red Outline which is the outline of the drawing sheet at the CRT. The Blue area represents the area currently displayed at the CRT.

1.4 COMMAND MENU WINDOW

The Menu Window (lower left corner of the CRT) displays either the major routines, their subroutines, or options as used to control the system during the design activities.

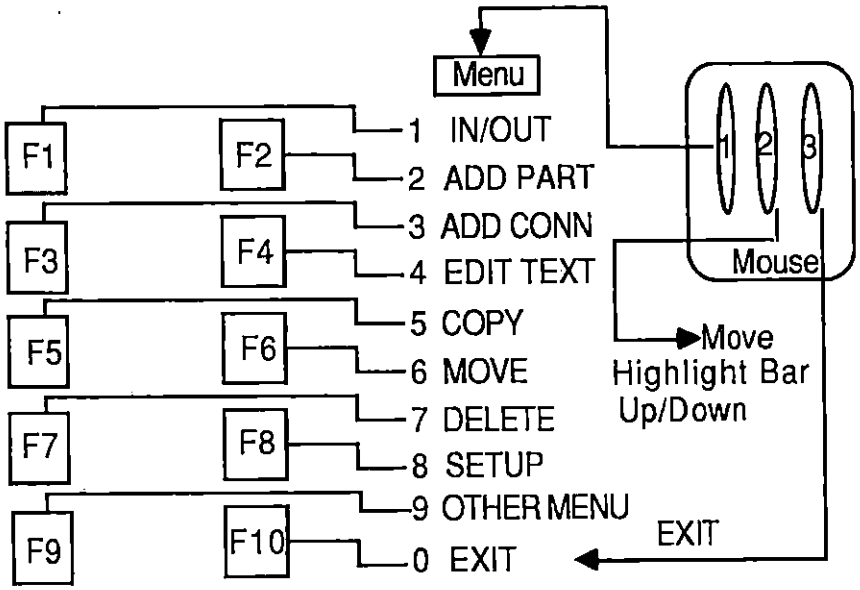
The lower left window of the CRT contains the Command Menu. The top of the window contains the title block indicating the routine or subroutine that you are currently using. Below the Title block are names of the subroutines which, in turn, are associated with the Function Keys 1 through 10.

This multi-line Title window is used in conjunction with the Menu Command Window and identifies the selected major command routine title, its first subroutine, and then the next lower level or option being developed. For example, if in the Major routine of IN/OUT, and you were calling in a file, the Title window would display:

IN/OUT
JOB/IN

The Title window changes with changes of the Command Menu.

1.5 COMMAND MENU SELECTION



Function Keys

MENU COMMAND SELECTION

The Command Menu items are selected with either the "F" Function Keys located on the keyboard, or the mouse buttons.

Menu routines, subroutines and options 1 through 10, are selected with the F1 through F10 buttons located on the keyboard.

The left-hand mouse button (Button #1) is mapped to select or activate the routine, subroutine or option over which the Menu Highlight Bar has been placed. The middle or mouse Button #2 is used to move the Menu Select Highlight Bar over the menu items to be "selected." Finally, the right-hand mouse button (Button #3) is mapped to the EXIT routine in the same fashion as the Function Key F10.

To move the Highlight Bar, hold down Mouse Button #2 and move the mouse Up or Down, and the Highlight Bar will follow. Let up on the Button #2 when the Highlight Bar is over the desired Menu. Alternately, move the cursor into the Menu Command Area, and the vertical movement of the mouse can be used to move the Highlight bar to diferent Command Menu positions.

A third way of moving the Highlight Menu Bar is by individual "Clicks" of Mouse Button #2. Using this method, a "Click" of Button #2 will advance the Highlight Bar from Top to Bottom of the Menu Commands, with each click advancing the Highlight one Menu item. To use this technique, you must change your Software slightly. Refer to Part III, Chapter One, Section 1.4.7

In summary, Function buttons F1 through F10 can be used to select any of the ten Command Menu choices. The mouse, in addition to controlling the cursor location, can also be used to select Menu Commands.

The designer will find that a position with the left hand over the Function buttons and the right hand on the mouse is the most comfortable operational position; and with practice, the simultaneous selection of the proper routines and the control of the cursor will become rapid and efficient. Alternately, the designer can use only one hand and the mouse to position the cursor, as well as to select the Menu item. It is suggested that you attempt both methods and utilize the technique best suited to your personal reaction.

1.6 SYSTEM GRID VS. DOT GRID

You need to understand about the System Grid. The ***PADS-LOGIC*** database has a System or Snap Grid varying from 2 Mils to 2,000 Mils, in 1 Mil increments.

For your use, a DOT grid is displayed as white dots when the CRT presentation is expanded to the 2nd level and above. This DOT grid is simply for your visual convenience while doing graphical work. The System Grid, however, is a "Snap" grid that, when moving or placing a part or a conductor will result in the part or line being snapped to the "SNAP" or System Grid when placed.

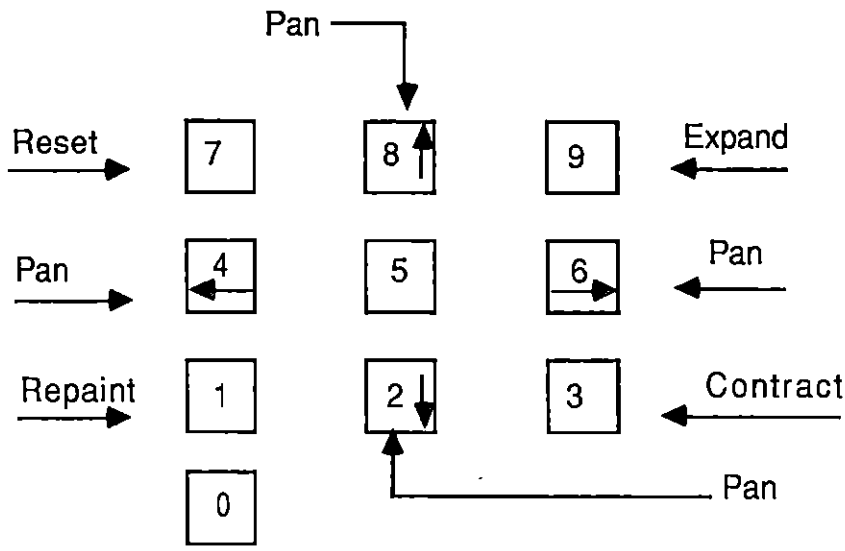
Let's see how the Snap Grid works while placing a Part. Have the DEMO file displayed at the CRT, as discussed earlier. Place the mouse cursor over the outline of any part and Select F6 MOVE. The part is attached to the mouse, and will turn White. Now you can move the component. Move the Component around and notice that it moves in 100 mil increments and that you can move the Component and place it on 100 mil increments (assuming the Snap Grid is set to 0.100"). The same conditions apply to Conductors, or other items. Notice the X/Y position is displayed in the White Window, and moves in 100 mil increments, as you move the part.

NOTE: To return the part to its original location, select F0 EXIT.

1.7 CRT MANIPULATION

CRT manipulation is the means by which the designer moves the drawing presentation around on the CRT, locates it in different areas on the CRT, expands the CRT presentation, contracts the CRT presentation; and in general, manipulates the CRT for the activity in process.

Located on the keyboard is a Numeric keypad containing Numeric Keys 0 through 9. Nine of these Numeric Keys, numbers 0,1,2,3,4,6,7,8 and 9 (Not 5) are used for manipulating the CRT, as shown in the following diagram:



CRT MANUPULATION KEYS

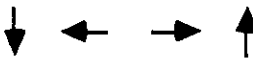
1.7.1 REPAINT - NUMERIC KEY #1 (END)

Numeric Key #1 (END), is used to REPAINT the CRT screen without Expanding, Contracting or Moving the contents. Repaint is used to "fill in" blanks in the image at the CRT raster after parts/conductors have been deleted or moved.

1.7.2 RESET - NUMERIC KEY #7 (HOME)

Numeric Key #7 (HOME), is used to reset the sheet drawing center to the center of the CRT and reset the Zoom scale to 1. Use Key #7 when you have lost your orientation on a large Drawing Sheet. Try it now.

1.7.3 PAN - NUMERIC KEY 2, 4, 6, 8



These keys are used to move or pan the presentation at the CRT

around to different locations of the CRT. The process of PAN does NOT enlarge or contract the scale.

To PAN, press Numeric Keys 2,4,6 or 8, depending upon the direction you wish to PAN. The scale will remain unchanged, but the feature at the CRT will be repositioned. Try Panning at Expansion Level Three or four.

1.7.4 AUTO PAN

An alternate method of Panning the display is called "AUTO PAN." Simply move your cursor to the Top, Bottom, Right or Left edge of the CRT and the edge of the screen under the cursor will move or "Autopan" toward the center of the Display. Try Auto Pan now.

NOTE: In order to AUTO PAN, "PAN" must be enabled in the SETUP, PARAMETERS menu.

1.7.5 EXPAND - NUMERIC KEY #9 (PG UP)

This key is used to EXPAND or double the scale (decrease the area) of presentation at the CRT by a factor of 2, and centered on the current location of the cursor. To expand, place the cursor at the area to be expanded. Press Numeric Key #9, and the item at the cursor location will be moved toward the center of the CRT; and the presentation will double in scale.

1.7.6 CONTRACT - NUMERIC KEY #3 (PG DN)

This key is used to CONTRACT or reduce the presentation at the CRT by one half, centered on the current location of the cursor. To use Contract, place the cursor at the area to be contracted. Press Numeric Key #3, and the scale will be reduced in size by one half (more data presented at the CRT).

1.7.7 CENTERING - NUMERIC KEY 0 (INS)

Key 0 INSERT is used to move the area of the display that is under the cursor to the center of the CRT.

1.7.8 **STOP CRT REPAINT - (ESC)**

The Escape Key is used to stop the CRT Repaint during a long repaint.

1.7.9 **CREATING A BOX (DEL)**

To create a box that will be enlarged to fill the CRT screen, select Key DEL. In response to the menu, use F1 START and the Mouse to create a Box to enclose the desired area. Use F1 COMPLETE and the Box will expand to fill the CRT.

1.7.10 **COMMAND MENU MACROS**

The use of repeated selections of **EXIT** to move from deep within the menu structure can be avoided by special Menu Macros, provided with the system.

1. Go to the Top of the Schematic Menu Command (ALT T)
The simultaneous use of the ALT Key and the T Key will move the menu status to the Top Menu Level of **PADS-LOGIC**.

2. Go to the Top of the Library Menu Command (ALT L)
The simultaneous use of the ALT Key and the L Key will move the menu status to the Top of the Library Menu.

3. Go to DOS (ALT X)
The simultaneous use of the ALT Key and the X Key will Exit the **PADS-LOGIC** program and go to DOS.

1.8 **PRACTICE ASSIGNMENT**

With a DEMO drawing sheet displayed at the CRT, exercise all of the CRT manipulation Commands; i.e., PAN, EXPAND, CONTRACT, CENTERING, etc., until you are comfortable with the CRT Manipulation Commands and the contents of the display at the CRT. When you are comfortable, then spend some time going through the menus. Start off with the top level menu showing IN/OUT, ADD PART, etc. Select IN/OUT and explore all of the subroutines associated with IN/OUT.

Let's change sheets on this multi-sheet schematic. Select F1 IN/OUT, then Select F5 SHEETS and Respond to Display Sheet 2. Repeat for Sheet 3; and finally, Sheet 4. Notice that in addition to selecting a Sheet, you can ADD Sheets, RENUMBER Sheets and DELETE Sheets.

You can have up to 256 Sheets in your Design.

1.8.1 PRACTICE: SETTING UP A NEW SCHEMATIC

In this part of Tutorial One, you will learn how to start a new drawing, set up the number of sheets with the proper size, and then how to save the job file to hard disk.

To build up your proficiency with ***PADS-LOGIC***, let's start from scratch. Turn off your PC, turn it back on and initialize ***PADS-LOGIC***.

A D-Size Sheet is displayed at the CRT. Let's make our New Schematic a C size. Select F8 SETUP, then F2 PARAMETERS. A RedText Window is displayed. The Second line (right-hand column) contains:

SHEET SIZE (A-E or A4-A0): D

This sets the system default to a D size. We wish to use a C size. Therefore, use the ENTER Key or Mouse Cursor to advance the cursor until it is over the "D," and then type C. Note that the Default is changed to a C. EXIT back to the Main Menu, and note your new Schematic sheet size is now a C size.

With ***PADS-LOGIC***, you can have any size drawing sheet from A - E, or A0 - A4 (International).

In the Purple window, notice:

SHEET 1/1

This indicates that you have set up only one sheet for your training file. At this time, let's add two more sheets. Select F1 IN/OUT,

Select F5 SHEETS, then F2 ADD-SHEET; and in response to the prompt line:

OK TO ADD SHEET [2] (Y/N)>

Respond: Y<CR>.

Note at the top left window of the CRT you now have Two sheets in the drawing. Repeat for new sheet #3, such that the drawing will have Three sheets. By the way, did you notice when the prompt line was asking for the Sheet number, the prompt is for the next possible sheet number? The number will always be the NEXT assignable sheet number. Simply respond Y, and the next assignable sheet number will be assigned to you. In ***PADS-LOGIC***, you can have up to 256 sheets.

1.8.2 SAVING YOUR WORK

You should have a good understanding now of how to start and size the sheets of a drawing. Incidentally, all sheets in a Design must be of the same size.

Let's review how to SAVE your work on hard disk.

PRACTICE: Select F1 IN/OUT, and the Menu Structure for IN/OUT is shown. To save your file to hard disk, use F2 JOB OUT.

The prompt message will ask:

SCHEMATIC OUTPUT FILE NAME (CR = CAE.SCH)

If you want to save the file using the name of "Your Initials," simply enter your initials followed by <CR>. Notice the activity of the hard disk indicating the file is being saved. Also note in the Yellow Window, the word "Working," indicating the disk is working at saving the file.

Your 3 Sheet "Blank" schematic will be saved with the name of your initials and the extension .SCH; thus GAM.SCH.

PRACTICE: Now let's get some practice calling up the Drawing called "Your Initials," previously made.

Turn your PC OFF, then ON. Bring up the ***PADS-LOGIC*** Application Software. Bring up the Drawing named with "Your Initials," using F1 JOB IN. Check that it is the drawing you previously made; (it is blank, with only the sheet outline and contains three sheets), and is at the CRT ready for additional work.

Let's expand the Drawing sheet. Select Numeric Key 9, PAGE UP, once, then again. Notice that the Postage Stamp feature shows the display at the CRT, and it is the center portion of Sheet #1.

This completes Tutorial One. You should feel comfortable with bringing in a new Schematic or saving schematic files, as well as manipulating these files at the CRT; and thus, you should feel prepared to specify tasks, as required by the selection of the appropriate Function Keys.

In Tutorial Two, you will consider all aspects of Placing and Editing parts on a sheet of your schematic.

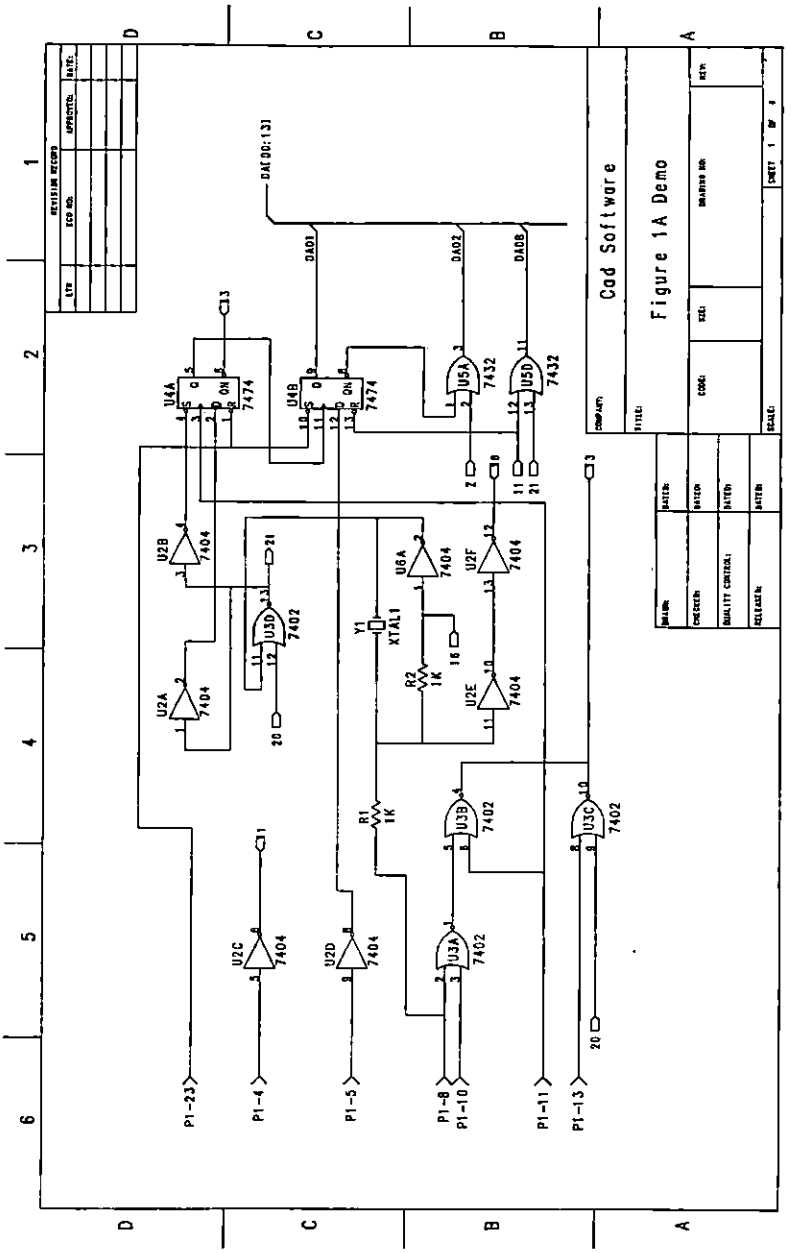


Figure 1A - DEMO, SHEET 1

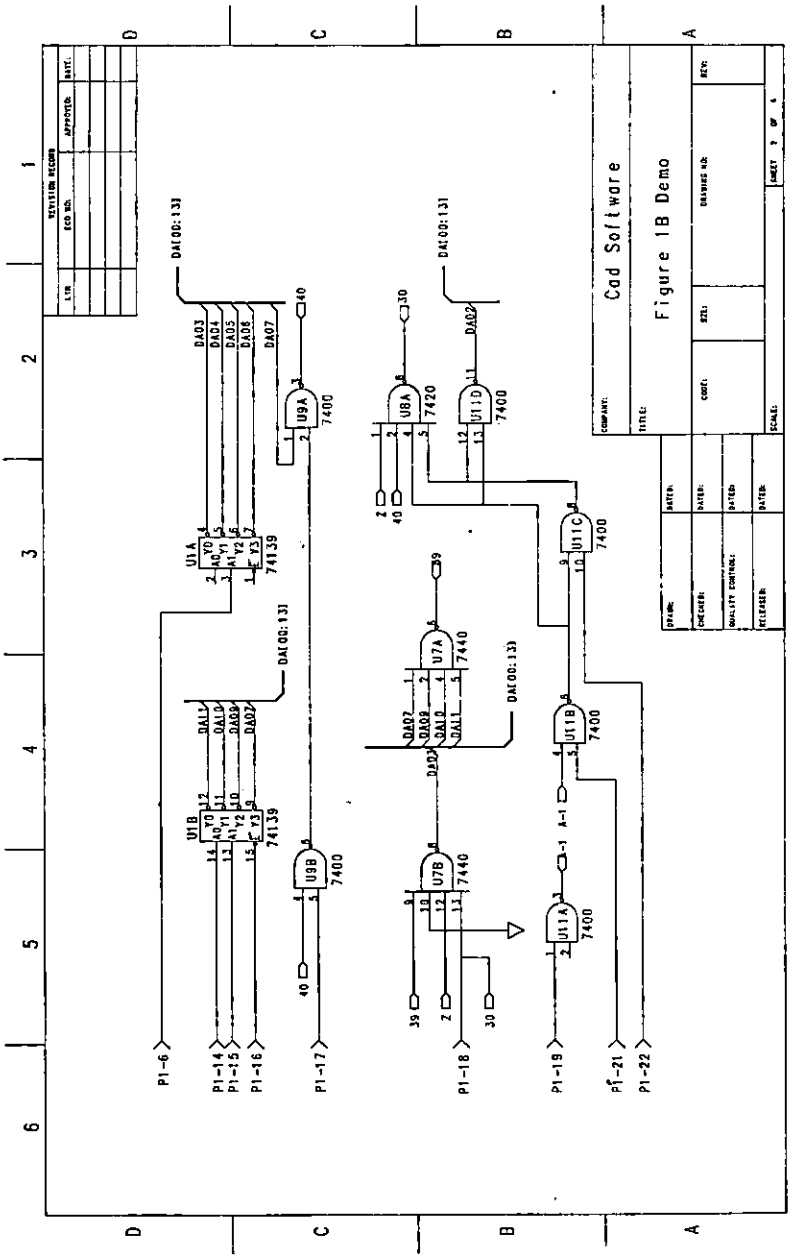


Figure 1B - DEMO, SHEET 2

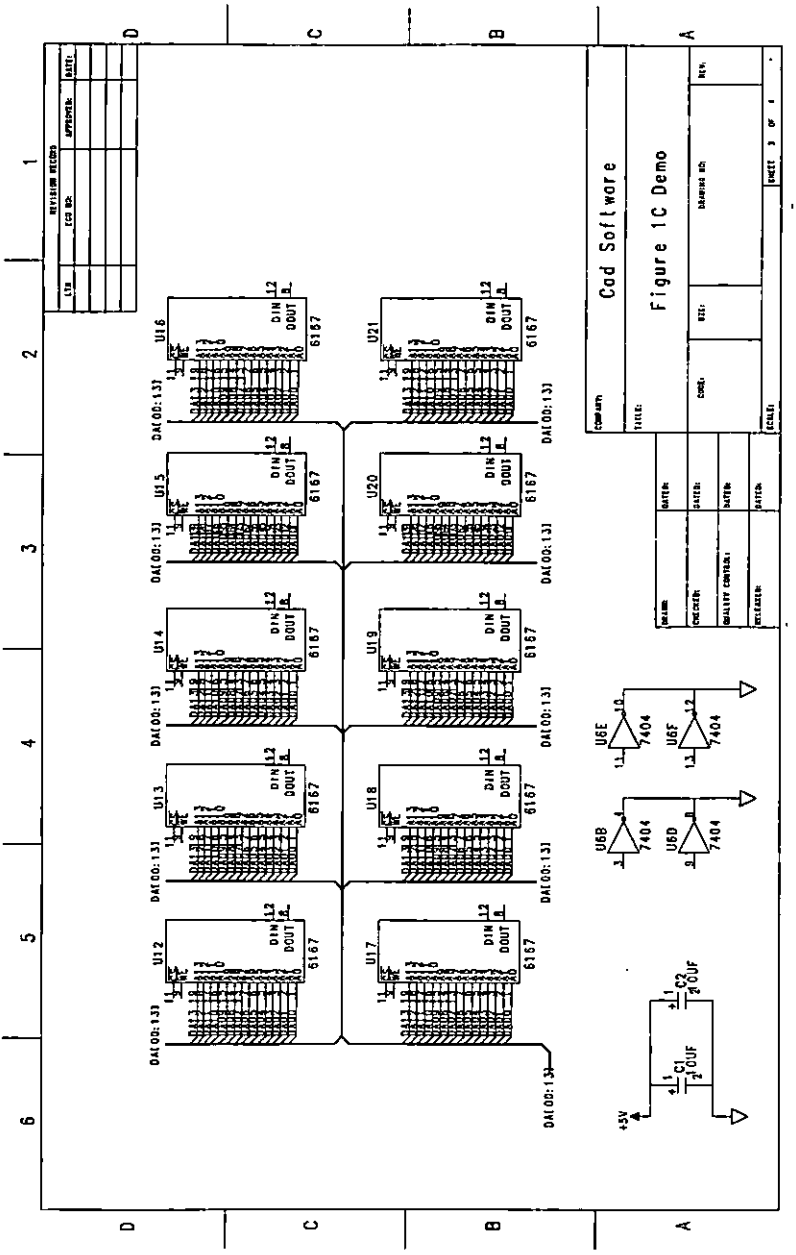


Figure 1C - DEMO, SHEET 3

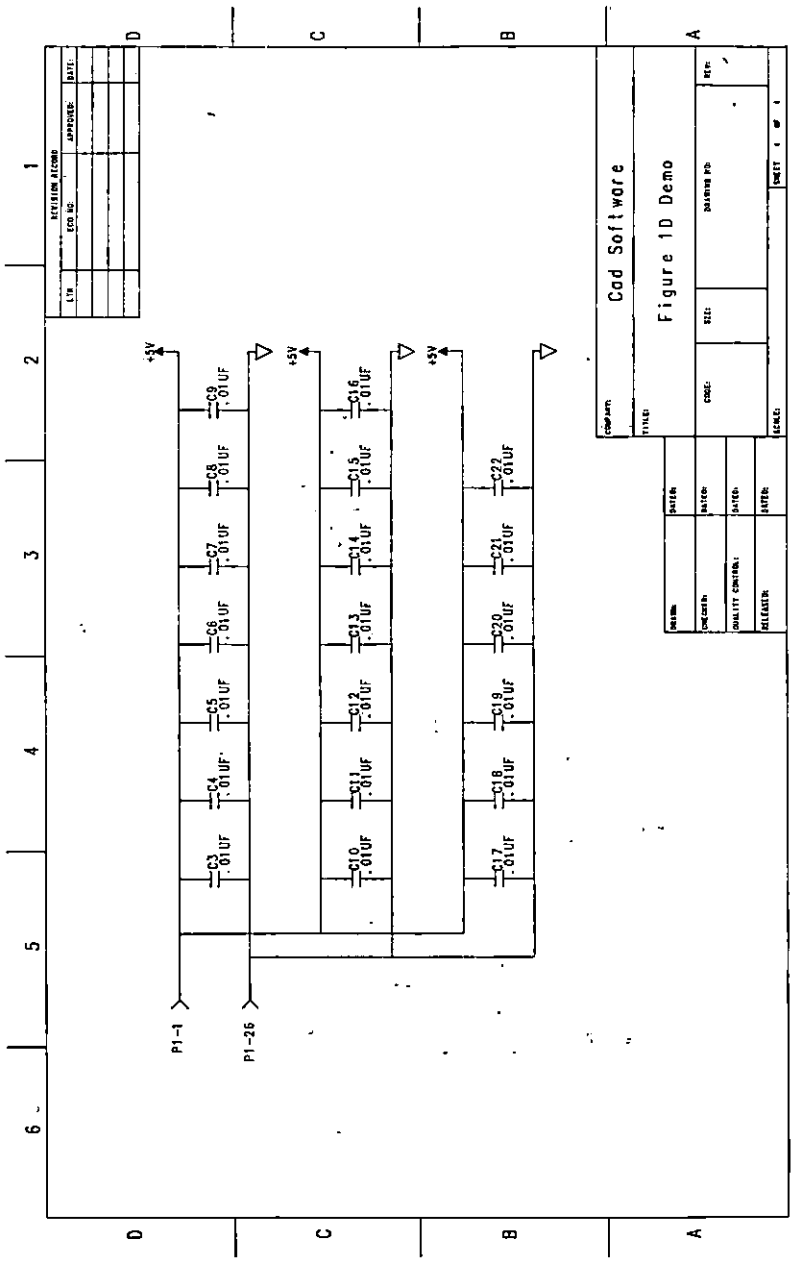


Figure 1D - DEMO, SHEET 4

TUTORIAL TWO:

PLACING PARTS ON THE SCHEMATIC

2.0 INTRODUCTION

This Tutorial is devoted to all the requirements for bringing up a part from the Library and placing it on the Drawing Sheet, manipulating it and presenting it for viewing at the CRT.

Before placing parts on your schematic, however, let's get an appreciation of the ***PADS-LOGIC*** Automatic Reference Designation assignment, while adding Parts to a Schematic.

You will need a Blank Schematic Sheet at the CRT. Let's get some practice using IN/OUT, call F1 IN/OUT, then F1 JOB IN. Call up the file you stored in Tutorial One, using your initials as the file name.

With a Blank Sheet 1 displayed at the CRT, call in a 1/8 watt resistor, as follows:

Select F2 ADD PART. You will be asked:

NAME OF PART TYPE FOR NEW PART

Respond with the Part Type of R1/8W <CR>. The Library will be searched for the part, and the Resistor will be attached to the cursor (in a White Color). Move the cursor around. The part follows. You may, at this point, place it in its location (Complete), Rotate it, mirror it in X & Y, and select its Alternate. Use F1 COMPLETE to place the part. Another part is immediately available, attached to the cursor.

Select F1 COMPLETE again. Move the cursor and select F1 COMPLETE, a third time. Then use the Right Mouse Button (F0 EXIT) to cancel the ADD PART command, and thus, add no further parts.

Place the cursor over the three resistors added above, and use

EXPAND (Numeric Key 9, Pg Up), until you can read the Reference Designations. Notice that the system has assigned the REF DES of R1, R2, & R3, with no effort on your part!

Interesting? What we have seen is the ***PADS-LOGIC*** REF DES Accounting module, that accounts for the REF DES on all the sheets in the Drawing and assigns the correct REF DES automatically. This ability to function across all sheets is a result of ***PADS-LOGIC's*** unique "Design-oriented" or "True Multi Sheet" Database. You don't need to keep track of or enter the REF DES. Let's see how it works on Multi-gated IC's.

Let's get acquainted with Autogate Reference Assignment on multiple-gate parts. Using F2 ADD PART, add an IC Part called, 74LS04. Note that when placed, and expanded (Numeric Key 9, Pg Up), we see the gate Decal and, without effort on your part, the Reference Designation of U1A has been assigned. Add another gate; it becomes U1B. Add the gate six more times; they become U1C through U1F, and then U2A and U2B.

In summary, Reference Designations and Gate Modifiers are assigned and accounted for by the system! You do not need to enter the data, and you do not need to keep track of the Reference Designations or the Gate Modifiers -- this is all done for you by the REF DES Accounting module. You concentrate on capturing the Logic of your design.

In the remainder of the Tutorial, you will place the Symbols representing the parts on the sheets of the Drawing called "SELFTRAN."

The Drawing that you will complete (SELFTRAN) is shown in Figures 2.1A, B, C and D (Rear of this Tutorial), and contains four sheets. Notice that it is the same drawing (DEMO) that you used to familiarize yourself with CRT manipulation earlier in Tutorial One, except that we have removed some Parts and Connections from each sheet of DEMO to create SELFTRAN. In your Self-training Tutorials, you will complete the four sheets of SELFTRAN, so as to match the Schematic DEMO shown in Figure 1A, B, C and D.

Take time out now, and examine both figures, DEMO Figure 1A, B, C and D, and SELFTRAN, Figure 2.1A, B, C and D. What you will be doing in the next several Tutorials is completing SELFTRAN (Figure 2), such that it is the same as Figure 1 (DEMO).

A word about DEMO and SELFTRAN: These drawings do NOT represent an actual circuit design. In fact, they are totally non-functional; instead, they have been developed so as to permit you to perform at least one of every function and activity required in creating a Logic Diagram or Schematic, thus giving you experience in performing any tasks you may encounter in creating a real Logic Diagram.

A comment about how you perform Parts Placement and the Connection Placement: Normally, you would create a Drawing sheet by placing some of the Part Symbols and connecting them up with (Connections) Conductors as you go, then more Parts and more Conductors until you have captured your complete design. This is NOT the way we will conduct the Tutorial Sessions on Parts Placement and Conductors. Instead, in this Tutorial, we will finish placing all the Part Decals on Sheets #1, #2, #3 and #4 of SELFTRAN. Having completed the placement of these Part decals, we will, in Tutorial Three, hook up the conductors connecting up the Parts. In other words, in Self-Training we CONCENTRATE ON A SPECIFIC FUNCTION at a time before proceeding to a different function.

NOTE: Before proceeding to complete this Tutorial, we suggest you turn to Chapter 3, Part III, ADD PART, and read this Reference material as "background."

PRACTICE: Let's proceed now with completing placing the parts on Sheet #1 of the SELFTRAN file. Using the knowledge that you have already gained, call up the SELFTRAN File. Select Sheet #1, and expand the Sheet. You are ready to start the placing of Part Decals.

We will now start placing the Parts on Sheet #1. Inasmuch as we will be working with Parts, select F0 EXIT, then F2 ADD PART. In response to the prompt:

NAME OF PART TYPE FOR NEW PART>

Respond with the Part Name of the part to be added. Type in the name of the part. Alternately, to browse the Library, use the wild card * or 74* (for 74XX IC's), and the Library will be listed in a pop-up window. Select the part using the cursor; the Library will be searched and the part will be attached to the cursor. The menu changes to the Add Parts Menu, as shown in the following:

ADD PART
1 COMPLETE
2 ROTATE
3 MIRROR X
4 MIRROR Y
5 ALTERNATE*
6 NEXT TYPE**
0 EXIT

*Only if Part Decal has Alternate Decal

**NEXT TYPE is used (when adding a part) in those instances wherein there are multiple type gates within the Part: an LM3900 for example. Let us say there were two types of gates--Gate A and Gate B. When the part is first used, Gate A will be brought up and added to the schematic. If you wish to continue using only Gate A on the schematic, then continue to ADD or COPY Gate A. If, on the other hand, you also wish to use Gate B, utilize F6 NEXT TYPE.

This menu contains the subroutines used for Adding, Rotating, calling the alternate (DeMorgan Equivalent), Mirroring and Placing the parts.

2.1 ADDING PARTS

You will now add Parts to Sheet #1.

In examining Sheet #1 of Figure 2.1A, we see a number of Gates associated with U2, a 7404 Integrated Circuit. Gates A through D are on the sheet. You are to add U2E and U2F as per Sheet #1 of Schematic DEMO, Figure 1a. Select F2 ADD PART, and you will be asked to define a PART NAME. Enter 7404 <CR>. The library

is searched for the 7404 and after a short search period, a gate is attached to the cursor. This is Gate E, of a 7404. Therefore, locate it approximately as shown in Figure 1A, Sheet #1 and release with F1 COMPLETE and F0 EXIT.

Enlarge the CRT and study the Gate Decal after you have released it. Note that it is just one gate of the 7404, and contains the Reference Designation U2E, together with the Gate Modifier, as well as the Part Type and Pin Numbers. From the Schematic of Figure 1A (DEMO), we see that Gate F is located near Gate E. Select F5 COPY. Move the cursor, with the next gate attached, near Gate E. Select F1 COMPLETE again, and the gate is placed. Use Mouse Button Three (F0 EXIT) to "cancel" a part from the cursor when all instances of the part have been placed.

NOTE: You used COMPLETE because the Part Decal was already attached to the cursor. If a new Part had to be obtained from the Library, you would have used F1 ADD PART. If the Part had already existed on the sheet, you could have selected COPY. To delete a part from the cursor after the desired part is placed, and no more parts of that type are requested, use Mouse Button Number Three.

HINT: You will want to see the entire sheet at one time to place the components. However, when the entire sheet is shown, the scale does not allow you to read the Reference Designations. Suggestion: Use the Keyboard Numbers 3 & 9 to Zoom the area of the sheet Up and Down, together with AUTOPAN, as required during the process of locating the gates.

U5 is a 7432. U5D is missing from Figure 2.1A. You are to add it. Use F5 COPY, and duplicate the gate using U5A. But notice! It is U5B, and you want U5D!!

You will now learn how to edit a REF DES assigned by the system. Place the cursor over the REF DES name of U5B, and Select F4 EDIT TEXT. The prompt line will read:

REF.NAME> U5B

Backspace to delete U5B and enter U5D <CR>. Click the Right

mouse button, and U5B will change to U5D at the CRT.

NOTE: The RED Pop-up window used when editing the text of the part is used for displaying or plotting invisible various text associated with a part and will be explained later.

U4 is a 7474. Copy U4B by placing the cursor over U4A and using F5 COPY, place the gate on Sheet #1 as per Figure 1A.
R2 is a 1K 1/4 Watt Resistor (R1/4W). Call up from the Library a 1/4W Resistor and place it as per Sheet 1A. We want the Resistor to be a 1K ohm part, therefore, go to F4 EDIT TEXT and turn on the VALUE: ??? Attribute Field by placing the cursor over the ???, and click Left mouse button. Next, using F4 EDIT TEXT, change the ??? to 1K <CR>, and select F2 THIS PART from the menu. Now hit <CR> twice. R2 will now have a Value of 1K. Repeat for R1 to make R1 a 1K Resistor.

We have now placed all the Parts on Sheet 1.

You should now create a JOB OUT file of the schematic you are creating. Do NOT use the Name SELFTRAN. You will overwrite the SELFTRAN files shipped with your system if you use SELFTRAN. Therefore, use SELF1 as the name of your Self-Training file, or some name other than SELFTRAN. SELF1 can be used at this time; then after you have done additional work, use SELF2, etc. In this fashion, you will have a copy of all files of the Schematic at different points of the design. Upon completion, you would, of course, delete all but the last file, which you would then SAVE for the future.

At this point, what remains to be done on your Training Schematic, is the location of Parts on Sheets #2, #3 and #4. Select Sheet #2 (using IN/OUT, F5 SHEETS, then F1 SEL SHEET or alternatively, use the Macro SH2 <CR>) and add the missing Parts on Sheet #2 so as to duplicate Sheet #2 of DEMO (Figure 1B). The missing parts are U1A, GND on Pin 13 of U7B, and Gate U9B (7400). Standard Ground is automatically generated on all IC's. You have added a Ground to Pin 10 of U7B, since in your Design Practice you may want to tie unused gates to Ground.

How do you add a Ground Symbol, or a Power Symbol, or an

Off-Page Symbol? Ground, Power and Off-Page Symbols are not added in the ADD PARTS routine, since they are NOT a part. Logically, they are a portion of a connection, therefore, they are added using F3 ADD CONN.

Let's do it now. Have U7B Zoomed in on the CRT. Place the cursor over Pin 10 and select F3 ADD CONN. Drag the cursor to the left and down and a White line will follow. When ready, select F5 GROUND, and the GND Symbol

will be attached to the connection. Let's assume you do not want to use the

Symbol; instead, you wish to use the

Symbol for Ground. Select F3 ALTERNATE and the

Symbol will be substituted. Select F1 COMPLETE to finish the connection and symbol (turns Blue).

Upon completion, make a JOB OUT file calling it SELF2.

Next, place the remaining missing Parts on Sheet #3 by adding U21 (6167). Edit the REF DES to make it U21.

Finally, we complete the Decoupling Caps on Sheet #4. To do this, we use F2 ADD PART and the Part Type Name of CK05. The Caps between C17 and C22 are to be as shown on Drawing Sheet 4.

NOTE: Because C1 through C17 and C22 have already been used in the Drawing, your first Decoupling Cap will be C18.

At this point, please go to Tutorial 10, read Section 10.3 JOB CONTROL, and start a Job Log as shown therein.

Make a file of your training file at this point, (SELF3).

What follows in this Tutorial is a miscellaneous group of features

associated with ADD PARTS with which you will need to be familiar.

2.2 ALTERNATE

ALTERNATE is a routine that allows the substitution of the Decal originally used for the Part and placed on the Sheet with an Alternate Decal (DeMorgan Equivalent), or different Decal representation. In order for this substitution to occur, the Part as stored in the Library must have an Alternate Decal, in addition to the Primary Decal.

ALTERNATE is located in the F2 ADD PART, F3 ADD CONN, F5 COPY and the F6 MOVE subroutines. However, the ALTERNATE menu will appear in these subroutines ONLY if the Part involved has an Alternate Decal. The menu is blank if the part does not have an Alternate Decal.

PRACTICE: Expand U1A of Sheet 1 so you can easily see the Decal. Place the cursor over U1A and Select F6 MOVE. The Decal will be attached to the cursor, and the menu will change. Select F5 ALTERNATE.

NOTE: The Decal name changes to its DeMorgan Equivalent. Use F1 COMPLETE.

2.3 COPY (PART)

Parts may be added to a sheet of the schematic by:

F2 ADD PART, or
F5 COPY

The F2 ADD PART subroutine is used when the part does not currently exist on the schematic and the part must be called from the library.

F5 COPY is used when the Part or a Gate of the Part is already on a sheet of the schematic.

To use COPY, place the cursor over the part to be "copied", and Select F5 COPY. Try it.

The part will be picked up, and a new part in White will be attached to the cursor. The Menu will change to:

COPY
1 COMPLETE
2 ROTATE
3 MIRROR X
4 MIRROR Y
5 ALTERNATE*
6 NEXT TYPE**
0 EXIT

*Only if Decal has Alternate Part

**NEXT TYPE is used (when adding a part) in those instances wherein there are multiple type gates within the Part: an LM390 for example. Let us say there were two types of gates--Gate A and Gate B. When the part is first used, Gate A will be brought up and added to the schematic. If you wish to continue using only Gate A on the schematic, then continue to ADD or COPY Gate A. If, on the other hand, you also wish to use Gate B, utilize F6 NEXT TYPE.

Perform the necessary activities, and then place the part on the location intended, using F1 COMPLETE.

NOTE: F2 ROTATE rotates the Part in either a 90 degree rotation from its original position, or back to the original position (not a full 360 degrees).

NOTE: MIRROR X or MIRROR Y "flips" or mirrors the part 180 degrees from the current position, referenced to either the X or Y axis, as selected.

PRACTICE: Copy a few parts to get familiar with COPY, ROTATE, MIRROR, etc.

2.4 MOVE (PART)

A Part Decal and its Text may be moved anywhere on the sheet. Place the cursor over the Part Outline, (not its Text), and use F6 MOVE.

The Part will change color to White and be attached to the cursor. The menu changes to allow Rotation, changing to the Alternate Part Decal (if the Alternate exists) and Mirroring.

The Part is placed using F1 COMPLETE.

When the Part is "picked," its data (REF DES, Part Type, Logic and Decal Name) is displayed in the Yellow Window. If connections are made to the pins on the Part, they will be moved orthogonally in X or Y, as the Part is Moved.

PRACTICE: At this point, practice moving a Part.

2.5 MOVE (PART TEXT)

The Text of a Part can be moved relative to the Part (except Pin Names) by placing the cursor over the Part Text and using F6 MOVE. The Part Outline and Text Outlines turn White, and the selected Part Text is attached to the cursor.

Movement of the cursor moves the Text with respect to its original position. The Menu changes to:

MOVE
1 COMPLETE
2 ROTATE
4 SIZE
0 EXIT

The Part Text may be Rotated, using F2 ROTATE. Each selection rotates the Text 90 degrees clockwise (Vertical), or 90 degrees counterclockwise (Horizontal).

F4 SIZE pertains to the changing of the Text Size. When selected, the following prompt appears:

CHANGE TEXT SIZE [HXXX WYYY]

With H being the Height of the current Text in mils and W being the Text Line thickness in mils. To change, enter XXX YYY <CR>. Accept the change with F1 COMPLETE.

NOTE: The Global Part Text Size for a Part Text category can also be changed using SETUP; then PARAMETERS.

NOTE: The Part Pin Numbers may be moved only a limited distance from their original location, so as to prevent confusing Pin Numbers with other Text.

PRACTICE: Try moving some Part Text. Change its Size and see that indeed the Text Size is changed at the CRT.

2.6 DELETE (PART)

DELETE is used to delete a part from the sheet. Place the cursor over the Part Outline or Part Text, and Select F7 DELETE. The Part will disappear from the CRT. F1 YES, in response to the Prompt Line, will complete the Part deletion. The mouse Button Number 3 will cancel the deletion.

A Part that is deleted with a connection attached will be deleted, along with the connection(s).

PRACTICE: Delete a Part and notice the results.

2.7 PART TEXT EDITING

There are various text attributes associated with a Part Decal. F4 EDIT TEXT allows the manipulation and editing of text associated with a Part Decal (as well as other text on the schematic).

To use, place the cursor over the text of a Part Decal (Expanded Scale), and Select F4 EDIT TEXT.

2.8 REFERENCE DESIGNATIONS

When the cursor is placed over the Part Reference Designation, three actions occur:

1) The Part Decal Outline becomes White (to indicate the part has been picked).

2) The picked REF DES is displayed on the Prompt Line.

3) A Red Pop-up window is displayed listing the Part Text.

To change the Part REF DES, use the backspace key to "erase" the REF DES displayed on the prompt line. Next, enter the new REF DES <CR>. When accepted, the REF DES will be changed. If not accepted (usually because it is already used in the Drawing), an error message is outputted.

To turn the Part REF DES OFF, from (1) being displayed at the CRT, AND (2) plotted at the plotter, use the Red Pop-up Window as a means of toggling the displayed REF DES ON and OFF. Place the cursor over the REF DES On/Off switch in the Pop-up window and use F1 SELECT to toggle OFF/ON.

Let's try it!

PRACTICE: Change U1A to U3A. This was not accepted because the system performed an "ON-LINE" check and found U3A already exists.

Now try U10A. This was accepted, of course, since U10A does not exist.

Now turn U1A OFF at the CRT (and from being plotted) by toggling the Part Text Pop-up window REF DES Line to the OFF position. Note that when the CRT is refreshed, the REF DES of U1A is no longer displayed (and will not be plotted). (This condition will prevail on all gates of U1, but not other components).

2.9 PIN (PAD) NUMBERS

When the cursor is placed over a pin number, and F4 EDIT TEXT is selected, the following prompt appears in the prompt line:

SELECT PIN TO SWAP WITH PIN XX.X

The intention of this feature is to permit you to swap the location of pins on a part decal from the location as contained in the Library. This allows you to change the flow of the conductors on a sheet of a schematic so as to make the flow a more attractive presentation.

In response to the prompt above, use the cursor to select the pin, whose location is to be swapped with the initially selected pin. The location of the two pins' symbols, numbers and names, are swapped on the part decal.

NOTE: The pin swap activity MUST be done and completed BEFORE connections have been connected to the pins. Swapping Pins after connections have been made to the Pins will not be performed and an Error Message will result.

Pin Names may be suppressed from being displayed at the CRT or plotted by being turned OFF using F5 EDIT TEXT and Editing the Red Pop-Up Window. Note the RED Pop-up Window is activated by selecting REF DES or PART TYPE, not PIN NAME.
NOTE: It is common practice NOT to display and plot the pin number of Discrete parts, such as 2 pin resistors, non-polarized capacitors, and other discrete parts, where pin numbers are not necessary.

2.10 PART TYPE

When the cursor is placed over the PART TYPE of a Decal, three actions occur:

(1) The Part Decal color is changed to White indicating the part is picked.

(2) The PART TYPE is displayed in the Prompt Window.

(3) The RED Pop-up Window appears, with the PART TYPE, highlighted (Blue). The Part Type can be turned ON/OFF from being displayed at the CRT and being plotted with this toggle.

The PART TYPE may be changed by backspacing the PART TYPE name on the prompt line, entering the new PART TYPE, and having the system accept the new PART TYPE name. In this regard, the new PART TYPE must: (1) be swappable in terms of number of pins, and (2) be in the ***PADS-LOGIC*** library. Obviously, a 14-pin Part cannot be replaced with a 2-pin Part or a 5-pin Decal cannot be replaced with a 2-pin Decal! A Part can be replaced with a Part with a Greater number of pins. However, if there are

connections already made to the Replaced Part, YOU should check the Replaced Part connections and their pins on the New Part.

If the "Change" is acceptable, the Menu will change to:

EDIT TEXT
1 This Gate
2 This Part
3 All Parts

Make the selection of your choice as follows:

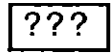
THIS GATE:
Only the gate selected will be changed to the new Part Type.

THIS PART:
All Gates of the part selected will be replaced with a gate of the new Part Type.

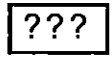
ALL PARTS:
All Gates of all Parts on all sheets of the design will be replaced with a gate of the new Part Type.

CAUTION! When Part Types are changed that have connections made to the pins of the changed Part Type, carefully Check, Repeat, carefully check that the resulting Connectivity is still valid. If not, correct it at this point, and before proceeding.

VALUE/TOLERANCE:
Some Parts, especially Discrete Parts, may have been created with a VALUE/TOLERANCE Attribute. Once created and the part is added to the sheet with these Attributes turned On, the Box



will be shown for either VALUE or TOLERANCE, or both. Use EDIT TEXT to change the



to the correct value. Then decide if ALL PARTS or THIS PART is to have the new Value.

PRACTICE: Using EDIT TEXT, practice changing Text associated with a Part.

2.11 SET UP (PART TEXT)

F8 SETUP is used to set up the display of Part Text at the CRT. When F8 SETUP is selected, the CRT SETUP menu is presented. Contained in the Setup Window are the following items pertaining to Part Text.

PARTS	ON	
REF DESIGNATOR	ON	BOX
PART TYPE	ON	BOX
PART TEXT	ON	BOX
PIN NUMBERS	ON	BOX

The left box for each item determines the color of the item if visible at the CRT. To change Text color, place the cursor over the desired color box in the Color Select Line at the top of the CRT, and use F1 SELECT to select the color. Then place the cursor over the left box of the Text item and use F1 SELECT. Upon Exit, the Text will be the color selected.

To make the Item Invisible, place the cursor over the same color box on the Color Select Line at the top of the CRT, as used for BAKGND. Use F1 SELECT, then place the cursor over the left box of the item to be made invisible and select F1 SELECT. The Box Outline will disappear and upon exiting to the schematic, the Item will be invisible.

Each Part Text item except Pin Name uses a box around it to show its location and true size when plotted (the Box is not plotted). This is represented by the Right Box with each Part Text item in SETUP. To turn the Box On/Off or change its color at the CRT, select a color from the color window at the top of the CRT (the same

color as the CRT background, will make the Box invisible). Then place the cursor over the Right Box and select F1 SELECT.

PRACTICE: Use SETUP to change the color of items associated with Parts, as well as to delete the Box associated with Part Text. Upon completion, return to the original Color/Box Setup.

2.12 SEARCH MACRO

A later Tutorial on Macros will fully describe the MACRO function. For now, while in ADD PART, learn to use SEARCH MACRO.

To Find a Part, select SEARCH MACRO by typing SXX.Y, Where S is "SEARCH" and XX.Y is the REF DES of the part, including the Part and Gate Modifier (if required) and Pin Number <CR>. The SEARCH MACRO will search all sheets, display the correct sheet, then identify the Part with the cursor.

At this time, in the use of this Tutorial, you have used or studied all the features available for adding parts to a sheet of the schematic. Complete this Tutorial, if you have not already done so, by placing all the Parts on all Four Sheets in the approximate location shown for the parts in Figures 1A thru 1D.

Study each sheet against Figure 1 to insure you have all parts placed.

2.13 PRACTICE: PLOTTING YOUR WORK

You will want to obtain a hard copy plot of each sheet for further study. Assuming you have a Matrix printer on your CPU, go to Tutorial Six, Section 6.2 and following the instructions for PLOTTING SHEET 1 contained therein; plot Sheet 1, followed by the remaining three sheets at a 1:1 Scale.

2.14 PRACTICE: SAVING YOUR WORK

When finished with Plotting, you must save your file for use in Tutorial Three, ADDING CONNECTIONS. Use F1 IN/OUT, then F2 JOB OUT and in response to the prompt asking for a file name, enter the file name you are using for this effort.

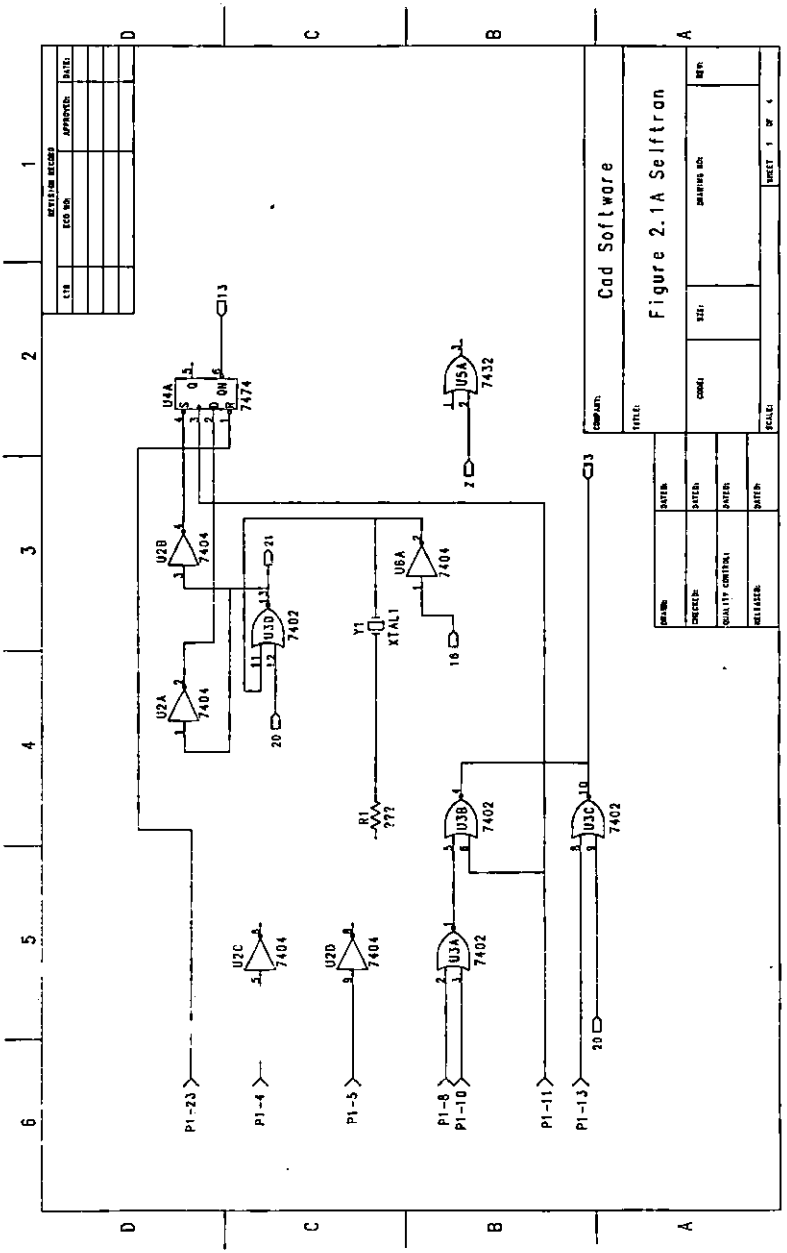


Figure 2.1A - SELFTRAN, SHEET 1

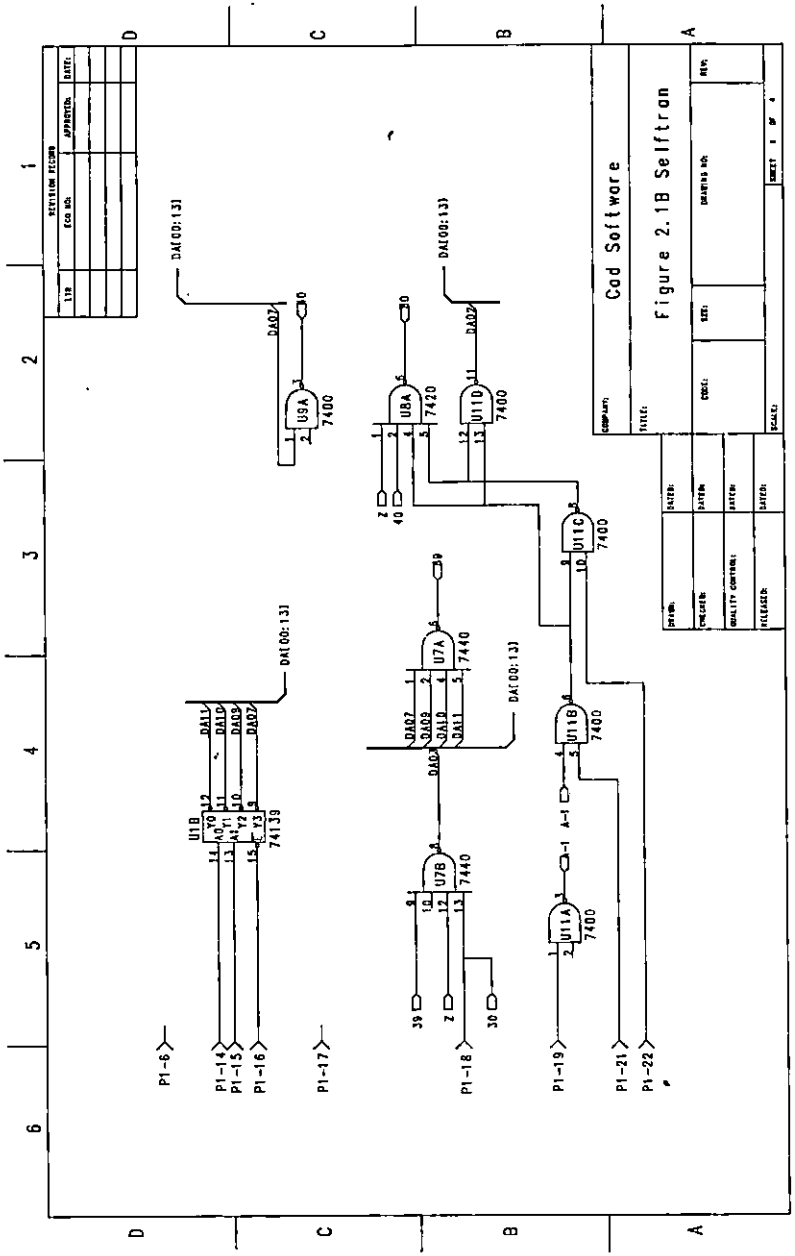


Figure 2.1B - SELFTRAN, SHEET 2

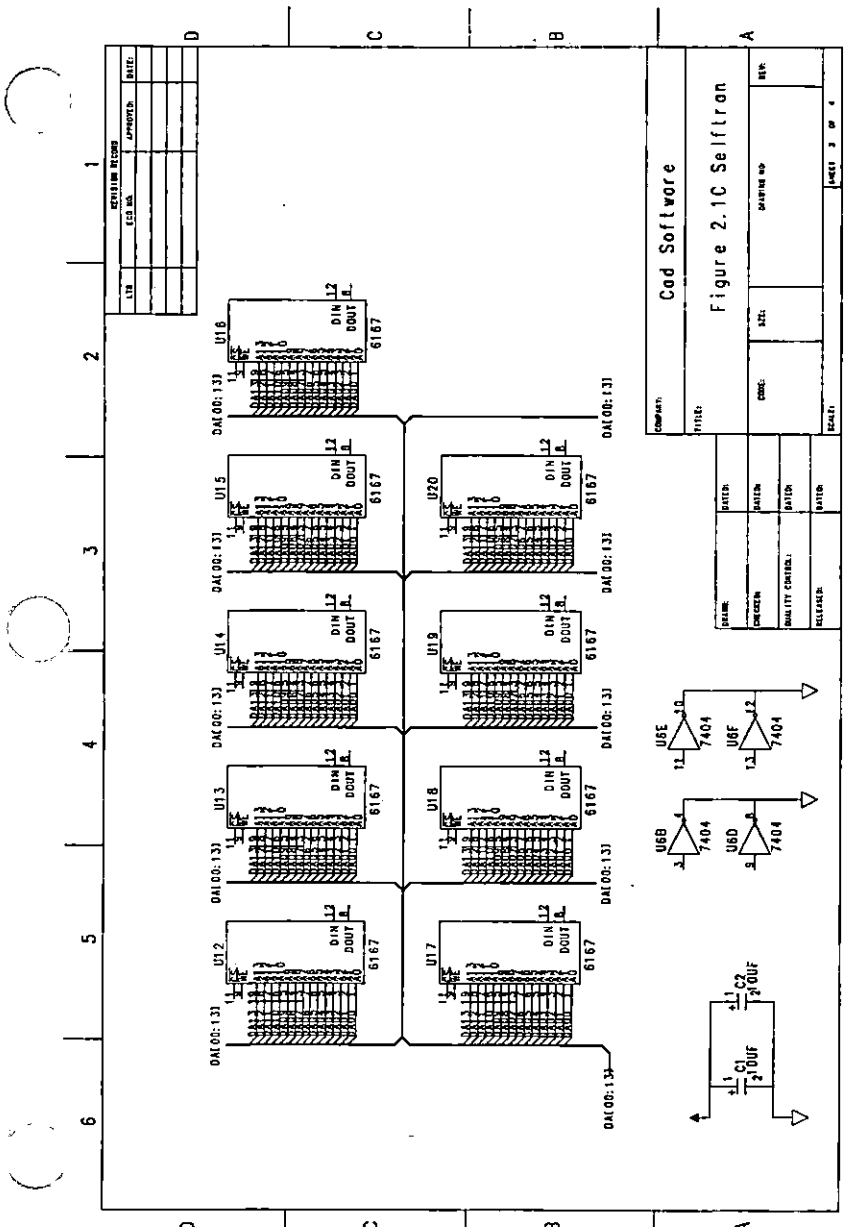


Figure 2.1C - SELFTRAN, SHEET 3

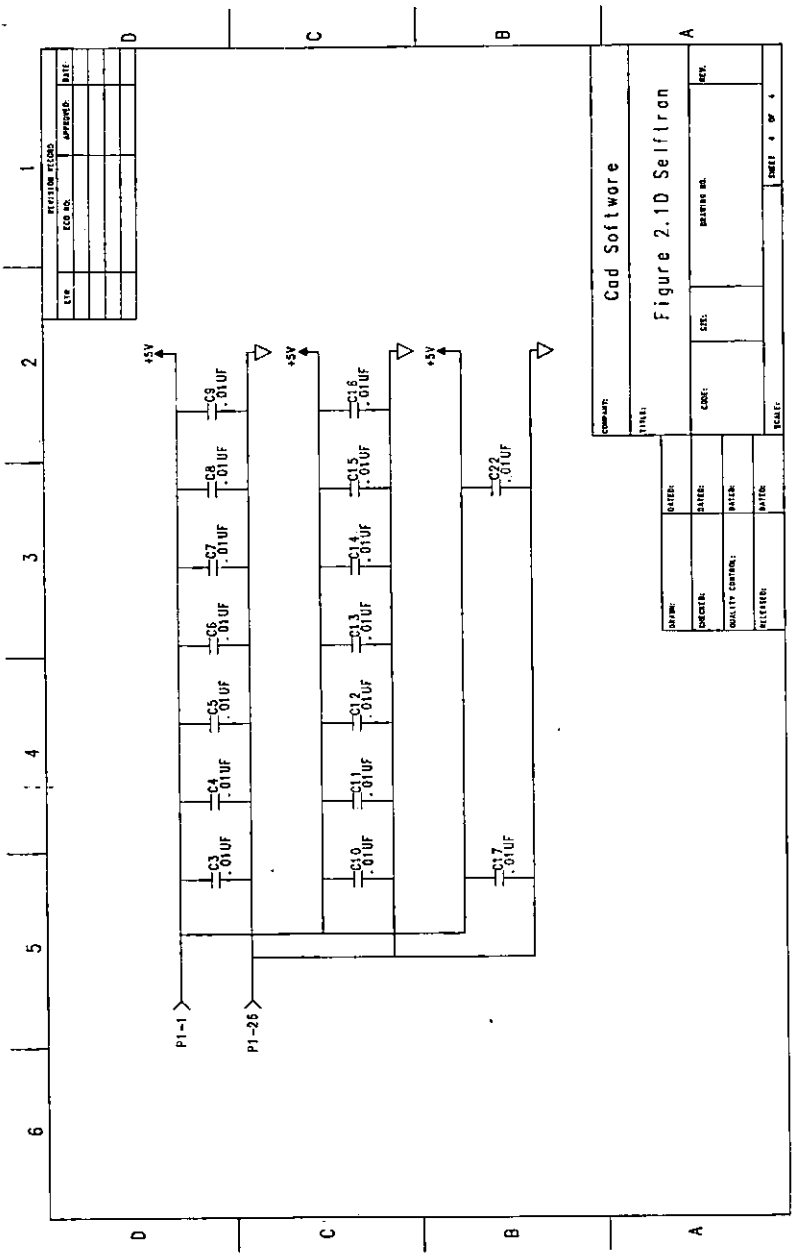


Figure 2.1D - SELFTRAN, SHEET 4

TUTORIAL THREE:

ADDING CONNECTIONS

3.0 INTRODUCTION

This Tutorial presents the features used to connect up the connections on the Schematic.

At the end of the Tutorial, you will have used all the features in the software required for this task, and will have "hooked" up all the Connections on the sheets of your schematic.

3.1 ADD CONN(ECTION)

F3 ADD CONN is used for adding connections to the schematic. Connections can be added:

1. From Part Pin to Part Pin
2. From Part Pin to another connection
3. From another connection to a Part Pin
4. From a connection to another connection
5. "To" a Bus, but not "From" a Bus
6. From a Part Pin or other connection to an Off-page tag
7. To a Ground or Power Symbol

NOTE: When starting or finishing on a connection, an automatic "TIE DOT" is inserted at the junction.

To add a connection, (1) place the cursor over the item from which the connection is to originate and (2) select F3 ADD CONN and drag the cursor in the desired direction.

A White connection will follow the cursor movement in one of the four orthogonal directions and the menu is changed to the following:

ADD CONN
1 CRN/FINISH
2 DEL CRN
3 ANGLE
4 OFF PAGE

5 GROUND
6 POWER
0 EXIT

F1 CRN/FINISH is used to (1) Bend the connection being hooked up and (2) finish the cursor at the Pin, Connection or Bus at the cursor location.

F2 DEL CRN is used to Undo the last corner.

F3 ANGLE is used after Corner to create a diagonal connection segment. When ANGLE is selected, this mode will continue until F1 CORNER is used again.

F4 OFF PAGE is used to put an Off-Page symbol that will tie the connection into another connection on a different page, using the same Off-Page Tag Name.

F5 GROUND adds a GROUND symbol to the connection and includes the connection in the Ground Net.

F6 POWER adds a +5V symbol to the connection and includes the connection in the +5V Power Net.

The Connection is displayed in White until connected at the opposite end, at which time the connection changes to Yellow.

To put in a connection, place the cursor over a Pin on another Connection and select F3 ADD CONN and start the connection. As you move the connection, put in a corner using F1 CRN/ FINISH. Notice you can "Bend" the connection to fit a pattern exactly as you wish. Notice a segment of the connection can be dragged in either an Up, Down, Right or Left fashion. If you make a corner and wish to take it out, use F2 DEL CRN. Use F4 ANGLE to put the connection on an Angle.

Hook up or finish the connection with F1 CRN/FINISH. You must have the cursor over a legal item to complete the connection (i.e., Pad, another connection, Bus, Off-Page PWR or GND Symbol).

3.2 PRACTICE: STARTING A CONDUCTOR FROM A TERMINAL

The most frequently used method of entering a Conductor is to start the conductor from a terminal on a Part. Let's do this now.

Have Sheet 1 of your SELFTRAN Schematic, with the Parts placed from Tutorial Two, displayed at the CRT. Let's practice putting in a conductor from a Terminal Pin, with the conductor starting from P1.4 and going to U2C.5. Place the cursor over the terminal associated with P1 Pin 4 and select F3 ADD CONN to start the conductor. Draw the cursor to the right--a White line will follow. Place the cursor over Pin 5 of U2C and select F1 CRN/FINISH. When the conductor has been made, the White line representing the conductor will turn to Yellow.

Try placing other conductors from U4B wherein the conductor starts from a Terminal and finishes on a Terminal. In order to finish or end on a Terminal, the mouse must be within the system "CAPTURE" distance of the Terminal. Use F1 CRN/FINISH to steer or place the Conductor as you wish it to be placed.

3.3 PRACTICE: FROM A CONDUCTOR

There are instances wherein you will wish to start a conductor from another conductor. To do this, place the cursor over the location on the conductor where the branch conductor is to start, and use F1 CRN/FINISH. Once the conductor has started from another conductor, a Tie-Dot is put in automatically. The completion sequence is the same as starting from a terminal.

Let's try it. On Figure 1A, go to the conductor running from P1.8 to U3A Pin 2 and locate the connection starting from that conductor and terminating on Resistor R1. We will now enter that conductor on SELFTRAN. Place the mouse over the conductor segment leaving P1.8. Select F3 ADD CONN and the White connection shows up following the cursor as it is dragged toward R1. Notice that Corner, Delete Corner, etc. are the same as if the conductor originated from a terminal. Place the cursor over Pin 1 of R1. Select F1 CRN/ FINISH. Notice that a Tie-Dot has been put in where the conductor is attached to the first conductor.

At this point, put in all the conductors on Sheet #1 that are missing except the conductors ending on the Bus named DA1[00:11], or ending in Off Sheet Tags.

Now go to Sheet 2 and put in the missing conductors except those tied into the Bus from U1A. We will put those in later.

3.4 USING OFF-PAGE (TAG)

F4 OFF PAGE tag is used to terminate a connection in order that the connection can be tied into a connection with the same Name on a different sheet.

Note that OFF PAGE may also be used to tie a connection to another Off-Page connection on the same page as long as the Connection Signal Names used by the two Off-Page Symbols are the same.

To use, while adding a connection , select F4 OFF PAGE when the cursor is at the location desired for the Off-Page Sheet. The last segment of the connection will turn WHITE and the following menu appears:

ADD CONN
OFF PAGE
1 COMPLETE
2 ROTATE
3 ALTERNATE
4 MIRROR X
5 MIRROR Y
0 EXIT

ROTATE permits the rotation of the Off-Page Symbol 90°.

ALTERNATE permits the substitution of an Alternate Off-Page Symbol.

MIRROR X/Y mirrors the symbol about its X or Y axis as appropriate.

COMPLETE adds the Off-Page symbol and asks for a Net Name.

You must assign a Net Name to the Net that is OTHER than the Net Name automatically assigned by the system and displayed in the Yellow Window. A White Box that will contain the Net Name will appear at the location of the Net Name. If you wish to Relocate the Name with respect to the Off-Page symbol, use F2 MOVE.

When finished, finalize the entry with F1 COMPLETE.

All connections on all sheets using the same Off-Page Name will be Automatically tied into a Single Net.

PRACTICE: Continue adding connections to all four sheets except those going to Buses.

At this point in the tutorial, you should have all the connections on all sheets connected except those tied to Bus Lines (Later). Check to see you do and make a file and SAVE.

3.5 MOVE (CONNECTIONS)

F6 MOVE is used to change or reconnect a connection after it has been connected or hooked up.

To use, place the cursor over the connection and select F6 MOVE. The connection will change color from Yellow to White.

Which segment of the connection you pick will affect the menu and what you can do while moving the connection.

If you pick either end segment of the connection (the beginning or ending segment), the connection will be disconnected from the pin, bus or other connection and becomes attached to the cursor. The menu changes to:

MOVE
1 CRN/FINISH
2 DEL CRN
3 ANGLE
4 OFF PAGE
5 GROUND
6 POWER
0 EXIT

You will note the menu is the same as when originally placing the connection and, in fact, this is exactly what you can do at this point. You can place the connection where you wish and reconnect to another pin or connection.

If you place the cursor over a middle segment of the conductor (other than a Diagonal), the connection will be picked and the menu will be changed to:

MOVE
1 COMPLETE
2 CUT SEG

The connection is NOT unconnected. You may reposition the picked segment (Right/Left (Vertical segment), Up/Down (Horizontal), and complete the movement with F1 COMPLETE.

F2 CUT SEG permits the dividing of the segment into two parts and moving either part.

PRACTICE: Use MOVE to relocate some connections. First, select a Middle Segment, then End Segments.

3.6 EDIT TEXT (RENAME CONNECTIONS)

F4 EDIT TEXT permits the user to rename the connection net. As a connection is entered into the database, a name is automatically assigned. The format is $$$ XXXX, where XXXX is an integer that is randomly assigned by the ***PADS-LOGIC*** Accounting Module.

1. A new unique Signal Name is automatically assigned by the system. To see the Net Name, look at the Yellow window while the connection is being connected. To see the Name after completion, place the cursor over the connection and select F4 EDIT TEXT; the name will be displayed in the Yellow window. (CRT must be at zoom level 3 or more.)

2. If a Net is started on an existing net, the new connection will use the name of the existing net to which it was connected.

3. If the Net is started as a new net (from a pin), and connected to an Existing Net, the existing Net will take the name of the new net.

If you wish to change the system-named net to a name of your choice, place the cursor over the connection to be changed. Select F4 EDIT TEXT. The connection will turn White, its present name is displayed in the Yellow window and the prompt message line will display:

NET NAME>

Type in the New Net Name <CR> and it will be displayed on the net. You may move the name to a different location using F2 MOVE. Enter the Net Name with F1 COMPLETE.

NOTE: Connections to a Bus have their names also displayed on the Prompt line.

NOTE: User assigned net names will be displayed at the CRT and plotted on the Drawing.

PRACTICE: Change the Name of some connections and observe the results.

3.7 DELETE (NET)

F7 DELETE is used to delete a connection from a net.

To use, place the cursor over the connection to be deleted and select F7 DELETE. The connection will disappear and the Net Name and connection End Pins will be identified in the Yellow window. To confirm the deletion, select F1 YES.

Note that with a single connection net, the entire connection is deleted; with a multi connection net, only the connection branch selected with the cursor will be deleted.

3.8 SETUP (CONNECTIONS)

F8 SETUP may be used to change the color of the connection as displayed at the CRT, or to make the connections invisible.

To use, select F8 SETUP. The Setup CRT display will be presented. The first entry is

CONNECTIONS [ON]

The Box Outline is displayed in the color used to display its color. To change the color of connections, select the new color from the color palette at the top of the CRT using the mouse, then place the cursor on top of the Connections Box and click the left button. The Box Outline will change color to the new color and when the connections are displayed, they will be in the new color.

To make the connections Invisible, place the cursor over the color in the color palette used as the background CRT color (usually black) and select this color. Place the cursor with the background color selected over the Connections Box and click the left button. The Box will disappear and the Connection Box will change to OFF, and the connections will be Invisible at the CRT display.

PRACTICE: At this time, use all the features described in this Tutorial, including MOVE, EDIT TEXT, DELETE, etc., until you are comfortable with them. Then to complete the Tutorial, if you have not already done so, connect ALL the Signal connections on all sheets of your SELFTRAN file EXCEPT Buses and the connections going to the Buses.

This will complete the tutorial on Adding Connections.

TUTORIAL FOUR:

ADDING BUSES

4.0 INTRODUCTION

A Bus is used to symbolize a group of connections containing closely related data such as exists in a Memory array. Entering a Bus into a schematic is somewhat similar to entering a connection, however, there are differences which you must understand.

Buses can be Added, Named, Moved, Modified, Connected to and Deleted.

To start a Bus, move the cursor to the location where the line is to start and select F9 OTHER MENU. The first subroutine of F9 OTHER MENU is F1 ADD BUS.

When F1 ADD BUS is selected, the Bus is started, in a White color, at the cursor location. The menu changes to:

ADD BUS
1 ADD CRN
2 DEL CRN
9 COMPLETE
0 EXIT

Dragging the Bus with the cursor will construct the Bus, including changes of direction of the Bus. F1 ADD CRN will add a corner that consists of 2 corners set at a 45° angle. F2 DEL CRN will delete the last corner entered. F9 COMPLETE will complete the Bus at the current cursor location. The Bus will turn Yellow when completed.

Immediately after F1 COMPLETE, a necessary part of entering a Bus is Naming the bus. The following prompt line appears:

BUS NAME (FORMAT = XXX [nn:mm])>

Enter the Bus Name at this point. The format is as follows:

XXX = the Bus Stem, i.e., DATAIO
nn = Lowest Numbered Bit contained in the Bus.
mm = Highest Numbered Bit contained in the Bus.

nn:mm is used for automatically assigning Bit Numbers to connections as they are being connected to the Bus.

An example of a proper Bus Name is DATAIO [00:07]

This will result in the Bus Bit Number of 0-7 being assigned automatically in sequence, as the connections associated with the 8 Bits of the Bus are connected to the Bus.

Select F1 COMPLETE to complete the Bus.

Using the same Bus Name on different sheets will tie the Buses with the same names together.

4.1 MOVE (BUS)

F6 MOVE may be used to rearrange a Bus, if used BEFORE connections are connected to the Bus. (AFTER connections are added to the Bus, GROUP is used to rearrange the bus pattern, as well as lengthen the bus. GROUP will permit the "grouping" of both the bus and the connections so as to be moved with a single movement.)

To use MOVE to rearrange a Bus without connections, place the cursor over the Bus and select F6 MOVE. The Bus will change to a White color and the menu will change to:

MOVE
1 COMPLETE
2 CUT SEG

These two commands, together with movement of the cursor, permit the complete rearrangement of the bus pattern.

4.2 ADDING CONNECTIONS TO A BUS

Connections may be connected to a bus that has been named. To

connect to a Bus, use the F3 ADD CONN command and start a connection on a pin or other connection and complete the connection by connecting to a Bus, NOT from the Bus to a Pin or other Connection.

When completed on a Bus, the resulting Net, including the resulting Net Name, is displayed on the prompt line, thus:

NET NAME>DATA00

This indicates Bit 00 of Bus DATA. The next connection attached to the Bus will be named DATA 01, etc. Bit Numbers will be AUTO INCREMENTED until the maximum Bit number in the Bus Name is reached, after which additional connections will be recycled to Bit 00. The menu changes to:

ADD CRN
CRN/FINISH
1 COMPLETE
2 MOVE
3 SWAP ANGLE

F2 MOVE permits the relocation of the Connection Name.

When a connection is attached to a Bus, the attachment is at an angle. The selection of F3 SWAP ANGLE changes the "approach angle." The distance of the Angle on the connection from the Bus, can be changed using SETUP, PARAMETERS.

F1 COMPLETE completes the adding of the connection to the database.

Note that to connect a connection to a Bus, the approach must be Orthogonal to the Bus segment, NOT on a Diagonal.

A Bus or Bus Connection Name may be changed using F4 EDIT TEXT. On Line Checking prevents illegal names.

PRACTICE: At this point, on Sheet #1 of SELFTRAN, create a Bus called XYZ [00:07]. Add connections to the Bus until you are comfortable with adding Buses and adding connections to the Bus. (Delete when finished.)

4.3 GROUP (BUS)

A Bus with connections added to it may not be MOVED. Instead, the subroutine of F4 GROUP, found in F9 OTHER MENU, is used. In this fashion, using MOVE, the bus and its connections may be moved.

To use, select F9 OTHER MENU and F4 GROUP. The menu will change to:

GROUP
1 DEFINE GRP
2 PASTE GRP

Place the cursor at the corner of an areas to include the Bus and its connections that are to be moved and select F1 DEFINE GRP. Drag the cursor so as to encompass the group and select F1 COMPLETE. The menu changes to:

DEFINE GRP
1 MOVE GRP
2 COPY GRP
3 DELETE GRP
4 CUT GRP
8 ADD ITEM
9 DEL ITEM
0 EXIT

Select F1 MOVE and move the Group containing the Bus and its connections to the desired location and release with F1 COMPLETE. You will find that if you place the cursor in the lower left corner of the group, and moving the Group SLOWLY, this will produce the best results.

4.4 DELETE (BUS)

F1 DELETE may not be used to delete a Bus to which connections

are tied. For this purpose, use the F3 DELETE GRP subroutine of GROUP.

PRACTICE: Finish all the connections and Buses of SELFTRAN by first adding the Buses and Naming them as per Figure 1A through D. Next, add the connections. Next, make a plot of all Sheets (A through D) containing the Buses. Study the sheets. Make any changes that are necessary and SAVE the file.

TUTORIAL FIVE:

ADDING 2-D LINES AND TEXT

5.1 2-D LINES

2-D LINES are used in the schematic for making any lines other than connection lines. Examples of such lines are Dimensions, Outlines, Charts, Jumpers without Connectivity, etc.

The 2-D LINE menu is located in F9 OTHER MENU. Select F9 OTHER MENU, then select F3 2-D LINES. The menu will change to:

2-D LINES
1 NEW POLY
2 NEW CIRCLE
3 MODIFY
4 COMBINE
5 COPY
6 MOVE
7 DELETE
8 FROM LIB
9 TO LIB
0 EXIT

When using 2-D LINES, remember to check the Default value for Grid and Width. You probably will want to use a Grid resolution of 10 and a Width appropriate for the Draw Line being created.

NOTE: The Width of the 2-D Line, in mils, is displayed in the Purple window. To change the Width, use the Modeless Command of WXX, where XX is the desired Width in mils.

NOTE: The 2-D Lines are displayed at their True Width if REAL WIDTH of SETUP, PARAMETERS is set to the 2-D Line True Width (mils).

You may want to use the X-Y location data at the top left of the CRT during drawing 2-D Lines. This data is of great assistance in accurately drawing specific length of lines.

5.1.1 NEW POLY

To start a new polygon, be in the OTHER MENU, ADD 2-D LINES option, place the cursor exactly where you wish the drawing line to start and select F1 NEW POLY.

The Menu changes to that of NEW POLY, ADD CRN. A White line is drawn starting from the cursor position and is displayed at the CRT. Use the options of F1 ADD CORNER, F2 DEL CORNER, F3 ANGLE and F8 END to create the drawing line, and if it is to be a closed polygon, when in the vicinity of the start point, select F9 COMPLETE for Auto Complete of the polygon. If the drawing line is to be ended prior to returning to the Start point, use F8 END to terminate the line at the current location of the cursor.

5.1.2 CREATE NEW CIRCLE

To create a circle rather than a line or polygon, be in the 2-D LINES subroutine, and select F2 NEW CIRCLE. Place the cursor on the circumference of the circle to be created (the start point) and select F2 NEW CIRCLE. Move the cursor away from the start point. The cursor movement distance and direction determines the circle diameter and its direction from the start point.

F3 ANGLE permits the direction of the circle diameter to be created at any angle, down to approximately 1^{o} from the start point.

When the circle is at the correct diameter and at the correct location, enter the circle into the database by selecting F1 COMPLETE.

PRACTICE: Try putting a Straight 2-D Line, a Polygon and a Circle on the schematic.

5.1.3 MODIFY (LINE)

MODIFY (LINE) is used primarily to change the pattern of the 2-D Line.

To use MODIFY, place the cursor over the 2-D Line and select F3 MODIFY. Note that when the 2-D Line and Line Type are selected, the Line Width and Line Type are displayed in the Yellow Window

at the upper left of the CRT.

F2 MOVE ARC permits changing a 2-D Line segment into an Arc and the changing of the Arc Radii. To use MOVE ARC, place the cursor over the 2-D Line Segment, select F2 MOVE ARC and drag the cursor for the distance the segment/arc is to be moved. Finish by selecting F1 COMPLETE.

F3 CUT SEG allows the cutting of a Horizontal or Vertical Line into two segments at the location of the cursor, and the subsequent movement of one of the resulting segments. F3 SWP CRN swaps the segments being moved. F4 LINE WIDTH permits the changing of the Line Width.

F6 LINE STYLE toggles the 2-D Line between a Dotted Line and a Solid Line. Place the cursor over the line to be modified and select F6 LINE STYLE. The Line will toggle between Dotted and Solid.

The Yellow Box in the upper left of the CRT will display this change.

NOTE: Dotted Lines are possible on Straight Lines only, NOT Circles and Arcs.

PRACTICE: Try Modifying the 2-D Lines and Circle previously created.

5.1.4 COPY (LINES)

To implement COPY, place the cursor over the Line to be copied and select F5 COPY. The copied drawing line is attached to the cursor, and it can then be moved by cursor movement and Rotated and Mirrored using the appropriate command.

5.1.5 MOVE (LINE)

The MOVE (Line) option is used to relocate or rotate the 2-D Line item.

To use MOVE, place the cursor over the Line to be moved and select F6 MOVE. The item will be attached to the cursor and it can then be

moved by cursor movement and rotated and mirrored using the appropriate command.

5.1.6 **DELETE (LINE)**

The DELETE option allows the deletion of a 2-D line from the database currently in memory. Note that if the line has been brought in from disk as a previously "saved" line, DELETE will only remove it from the schematic in memory and displayed at the CRT, NOT from the hard disk.

To utilize DELETE, place the cursor over the line to be deleted and select F7 DELETE. The line disappears and the message center provides the message:

CONFIRM DELETION Y/(N)?

A "Y" and Carriage Return will delete from the database; a "N" will cancel the Delete command, or F1 YES.

5.1.7 **COMBINE (LINES AND TEXT)**

COMBINE is used to group together into a single drawing item, a number of existing 2-D items as well as free text. After being combined, they can be manipulated (moved, rotated, copied, etc.) as if they were a single item, as indeed they are.

For example, you may want to Create Drawing Outlines/Charts (using 2-D LINE) and add free text, such as your Corporation's Name (Text), Combine them and Save them as an item in the 2-D portion of the ***PADS-LOGIC*** Library.

To use COMBINE, you should be in the 2-D LINE routine and select F4 COMBINE. The menu window will change as shown below:

COMBINE
1 ADD ITEM
2 DEL ITEM

To add an item to the group, select F1 ADD ITEM and the menu will

change to F1 SELECT. Move the cursor over the first line to be added to the group, then select it. The color will change to White, indicating it has been selected for combining. Move the cursor to the second line, use F1 SELECT and the cursor picks the second line and its color will also change. Continue the operation until all desired drawing lines and text are grouped or combined into a single group. After leaving COMBINE, the normal manipulation of a single Line such as Move, Rotate, etc., is applicable to the combined group of drawing lines.

NOTE: When combining Lines and Text, the Line item MUST always be selected prior to selecting the Text, otherwise an Error Message will result.

NOTE: When being combined, in order to identify the items being combined, 2-D Lines turn White and Text disappears until F9 COMPLETE.

DEL ITEM is used to delete a Line or Text from a group after the lines have been previously combined into a group. To use, select F2 DEL ITEM and in response to F1 SELECT, place the cursor over the item to be deleted from the group and use F1 SELECT. This line will be deleted from the group. The remaining lines within the group will continue to remain combined as a group unless also deleted.

PRACTICE: Create two to three Lines/Circles and some Free Text and use COMBINE to combine them, then Decombine.

5.1.8 FROM LIB (LINES)

The purpose of the FROM LIB option is to allow adding of "combined" lines and Text to the existing presentation at the CRT, a 2-D item that has been previously created and saved using F9 TO LIB. To implement, select F8 FROM LIB and, in response to the message:

ADD ITEM WITH NAME>

type in the name of the saved 2-D Line <CR> and a copy of the 2-D item will be transferred from disk into memory. FROM LIB is

a convenient way of recalling and using previously created, repetitively used 2-D Lines such as charts, dimension line arrows, etc.

NOTE: If you cannot remember the Name of the item, use the Wild Card * <CR> and a Red Pop-up window will appear listing all the 2-D Line items saved. Try it now and call up one of the items, say S: SIZEA.

5.1.9 TO LIB (LINES)

TO LIB is used to transfer to disk for storage, a 2-D line, or combined group of 2-D lines and text, that you expect to use repeatedly. Examples might be a Sheet Outline, dimension arrows or any drawing feature that has a long life expectancy. To use, place the cursor over the 2-D Line or combined group of lines and text and select F9 TO LIB and in response to the message:

SAVE ITEM WITH NAME>

type in the name used for saving the item. Any number of alphanumeric characters up to a maximum of 8 is valid. The 2-D Lines are stored in the Library PLIB in a subsection of the Library reserved for 2-D items.

PRACTICE: Save the Circle item you created. Call it CIRCLE. Now bring it back into the Drawing, using F8 FROM LIB.

5.2 ADD TEXT

ADD TEXT is a Text String Editor used to create free text on the schematic. When F2 ADD TEXT is selected in F9 OTHER MENU, the prompt line asks:

NAME>

Enter the text as follows:

1. Type the Text Entry and use <CR>. A White "Box" will be attached to the cursor. This Box is the true size of the Text string, when the schematic is plotted. Use this Box to position the text.

2. Check to see if the Text is at the correct Text Height and has the proper Line Width, and the Text Contents are as desired, in the Yellow window.

3. All alphanumeric and special characters on the keyboard are valid. The Text String may be 60 characters long, including spaces. (The Text has not been final placed on the board at this point.)

4. The Text Box will be attached to the cursor, AND the Menu window will change to as shown below. Move the cursor where the text is to be placed and use F1 COMPLETE to final place.

ADD TEXT
1 COMPLETE
2 ROTATE
3 SIZE
4 HORIZ JUST
5 VERT JUST
0 EXIT

F1 COMPLETE is used to "finish" the Text String entry. The Text is entered into the database according to the cursor location and at the indicated Text Height and Line Width. The Box that appears around the text is the actual size of the text when plotted. All text manipulation should be done before selecting F1 COMPLETE.

F2 ROTATE will rotate the Text string at its current location, 90° clockwise about the lower left corner of the Text string or 90° counterclockwise, each time ROTATE is selected.

F3 SIZE changes the Text Size from the current Default Text Size setting. The message:

CHANGE TEXT SIZE (HXXX WYY)>

appears. To change the Text Size, enter:

XXX YY <CR>

where XXX is new Height (in mils) and YY is new Line Width (in mils). The Text will be changed to the new Height and Line Width.

NOTE: Text may be at the CRT at its true width if the Real Width subroutine of SETUP, PARAMETERS is set such that the Real Width setting is equal to or less than the Text Line Width. Example: Text Line Width is 15 mils, set Real Width to 15 mils or less. You may need to change the scale at the CRT (Zoom In) to see the Line Width.

F4 HORIZ JUST and F5 VERT JUST will "Justify" the Text in the Horizontal (X) and Vertical (Y) direction respectively. HORIZONTAL JUSTIFY has three positions: Left, Center and Right. VERTICAL JUSTIFY has three positions: Top, Center and Bottom.

F0 EXIT is used to cancel a text string entered into memory and attached to the cursor (but NOT placed). If the text has been completed, F0 EXIT leaves the option, for the next higher command level.

PRACTICE: Put in your Name, using F2 ADD TEXT. Change Size.

5.2.1 COPY (TEXT)

The F5 COPY option is used to copy an existing text string and place it at any location. The COPY option is implemented as follows:

1. Place the cursor over the text string to be copied.

2. Select F5 COPY. The new copied text string will be attached to the cursor and can be moved to its final location. When the text string is copied, the menu is replaced with the following:

COPY
1 COMPLETE
2 ROTATE
3 SIZE
4 HORIZ JUST
5 VERT JUST
0 EXIT

3. Check the Text Size; change if required.

4. Move the copied Text into the proper location and select F1 COMPLETE.

PRACTICE: Copy "Your Name" and change its Size to H300 W20.

5.2.2 EDIT (TEXT)

F4 EDIT TEXT is used to modify a text string previously entered into the database.

To use EDIT TEXT, be in the top menu, place the cursor over the desired free text string and select F4 EDIT TEXT. The text string will disappear and the Box will turn White, indicating the Text has been picked. The selected Text will appear in the Prompt line. Use the Backspace key to erase; then enter the new string <CR>.

5.2.3 DELETE (TEXT)

DELETE is used to delete a text string previously entered into the Job database.

To implement DELETE Text, place the cursor over the text string to be deleted and select F7 DELETE. The text will disappear from the CRT and the following message appears:

CONFIRM DELETION Y/(N)?

A "Y" <CR> will permanently delete the text string from the database. A "N" will cancel the DELETE command.

5.2.4 MOVE (TEXT)

F6 MOVE TEXT is used to Move a Text string as well as to Rotate, Horizontally justify, Vertically justify, and change Text Size.

Notes on Adding Text

ADD TEXT is a powerful subroutine, allowing flexibility in size,

placement and rotation of text. The ADD TEXT subroutine may, at first, seem complex. You will find that after a practice session or two, it's use becomes quite easy. The following notes are intended to help you become familiar with text manipulation.

NOTE 1: When you have generated a text string, used <CR>, and the text doesn't appear at the CRT at the cursor, it probably is due to not setting the text level visible in the SETUP Display subroutine. Make sure Text is Visible.

NOTE 2: After repaint of the CRT, the text will appear at the proper height for the database scale currently in use. Consequently, if operating at a large board scale and text is entered, you may find the text that appears under the cursor to be too small to be readable. Change the CRT Zoom scale until readable.

NOTE 3: When using the ADD Text subroutine, make a habit of checking the Default Text Size before entering the text. If you select COMPLETE, the text will be entered at the default condition. To change the Size after the text string has been entered, use F6 MOVE, then the SIZE subroutine, and finally, F1 COMPLETE.

NOTE 4: If, upon running check plots, you find the text is the wrong size, use F6 MOVE Text and change to the appropriate new Size.

PRACTICE: Now practice adding a few strings of Text. Next, move the Text. Next, change the Size of the Text.

5.2.5 COMBINE (TEXT AND 2-D LINES)

You may use the COMBINE feature in 2-D Lines to combine 2-D Lines and Free Text. In the process, you must always combine Text with 2-D Lines, NOT the reverse. That is to say, the FIRST item selected when combining MUST be a 2-D Line.

PRACTICE: Try it. Create a 2-D Line polygon, a Circle, then some Text. Next, combine them together. Check that they are combined by moving them.

TUTORIAL SIX:

USING THE GROUP AND PLOT FEATURES

This Tutorial provides practice in the use of (1) the GROUP feature, and (2) the PLOTTING feature.

6.1 GROUP

The GROUP feature permits one to group together into a single entry, all or some of the items contained in the sheet of a schematic drawing, including the drawing sheet outline. After a Group is created, it may be Moved, Copied, Deleted, or Saved to hard disk for reuse, either within the same design, or a different design. Items can be added to the group or deleted from the group.

When a group is added to a schematic, the REF DES of the parts are automatically assigned.

A group can be created and used as a Hierarchical item.

To use Group, have a sheet of the schematic displayed and decide on the area to be grouped. Let's use Sheet #1, SELFTRAN, and Group the area in the lower left corner, containing P1.8, P1.10, P1.11, P1.13, U3A, U3B, U3C and Off-Page Symbols.

Select F9, OTHER MENU. Select F4 GROUP. The menu will change to:

OTHER MENU
GROUP
1 DEFINE GRP
2 PASTE GRP
0 EXIT

Place the cursor over one of the corners of the above indicated group of items, and Select F1 DEFINE GRP. Drag the corner diagonally to the other corner; a White rectangle will be created to enclose the Group Area. When the Group area has been enclosed, Select F1 COMPLETE to complete the Group Action. The GROUPED AREA will be displayed as a white color.

Upon selecting F1 COMPLETE, the menu will change to the following to allow you to further manipulate the Group:

OTHER MENU
GROUP
DEFINE GRP
1 MOVE GRP
2 COPY GRP
3 DELETE GRP
4 CUT GRP
8 ADD ITEM
9 DEL ITEM
0 EXIT

Decide on what action you wish to take before leaving GROUP. Otherwise, you will need to recreate the Group.

6.1.1 MOVE GROUP

F1 MOVE GROUP allows the moving of the Group about on the sheet of the schematic. Try it; note the old area of the group is blank. Note that connections outside the Group area are attached to the new location in an Orthogonal fashion.

6.1.2 COPY GROUP

F2 COPY GROUP allows the duplication of the Group, and relocation of the copy on the same sheet. Try it. Note the message:

CUT CONNECTIONS AT BOUNDARY (Y/N)?

The response to this question determines how the connections that are intercepted by the Group Outline will be handled.

A "Y" response will generally result in that portion of the connection inside the Group being included in the Group.

An "N" will result in the part of the connector contained within the Group not being copied, as an item within the new Group.

Including only a portion of a part within the Group Outline will

result in the entire part being included in the New Group.

Parts in the New Group will have REF DES Automatically assigned.

Refer to Part III, Chapter Nine, Section 9.4, "Notes on Group Connections," for the specific rules governing connections cut by a Group boundary.

PRACTICE: Use the GROUP feature to create a new Group and then copy it, paying particular attention to how Cut connections are handled, using both a Y and an N response to the "CUT CONNECTIONS AT BOUNDARY" Prompt.

6.1.3 GROUPING OF HIERARCHICAL SYMBOLS

The GROUP routine can also include a Hierarchical Symbol. When a Group is created that includes a Hierarchical symbol, a Warning message will be issued as follows:

WARNING: GROUP CONTAINS HIERARCHICAL SYMBOLS

You should be aware that when you include a Hierarchical symbol in a Group, the circuitry represented by this Hierarchical symbol will also be included in the Group and when pasted into another design, will include the circuitry associated with the symbol.

6.1.4 DELETE GROUP

F3 DELETE GRP allows the deleting of the GROUP from the database. After confirming the delete initiation, respond to the prompt:

CUT CONNECTIONS AT BOUNDARY (Y/N)?

in accordance with how you wish the Cut Connections remaining in the design, after Group Deletion, are to be handled.

A "Y" will result in the part of the connection remaining in the Design OUTSIDE of the Group area being renamed automatically,

with an Off-Sheet Symbol automatically assigned. You must tie the Net into the Design using F4 EDIT TEXT.

An "N" will result in the part of the connection OUTSIDE the Group Outline, being deleted back to a terminal pin, or the tie point to another connection.

PRACTICE: Create a new group, wherein some connections are interrupted by the Group. Next, delete the Group, after answering Yes to the prompt to "cut connections at the boundary." Examine how the cut connections remaining in the Design are named. Repeat, using an "N" response.

6.1.5 GROUP DELETE WITH HIERARCHICAL SYMBOLS

When a Group is being created for deletion of the Group, and this group includes a Hierarchical Symbol, the normal "Group Delete" message prompts are issued, along with a special prompt which follows:

DELETE ASSOCIATED SHEETS (Y/N?)

A "Y" response will result in the sheets represented by the Hierarchical symbol being deleted from the design.

An "N" response will delete the symbols but not the circuitry.

6.1.6 ADD AND DELETE

The Group contents contained within the Group Outline can be changed by adding items from outside the Group using F8 ADD ITEM. Items can be deleted from within the Group by use of F9 DEL ITEM.

PRACTICE: Create a Group on the sheet. Then, using F8 ADD ITEM, select some part that is outside the Group. Then F1 MOVE GRP. Note the added part from outside the Group is now a part of the Group and moves with the Group. Create another Group. Delete an item from within the Group, using F9 DEL ITEM. Move the group. Note the deleted item is not part of the Group.

6.1.7 CUT (GROUP)

A Group may be "cut" out of the Design, saved as a file, and "pasted," on another sheet of the Schematic, or used on another Design.

PRACTICE: Let's try it. Create a group on Sheet one. Select F4 CUT. Respond to the prompt:

ENTER GROUP FILE NAME (CR = CAE.GRP)

with a File Name of your choice.

Respond to the prompt:

CUT CONNECTION AT BOUNDARY (Y/N)

with:

Y

The file will be saved in the \PADS\LOGFIL directory using the name given to the file and the extension .GRP.

Connections cut at the Boundary during the cutting of a group use the same rules as GROUP COPY. Refer to Part III, Chapter Twelve, the Group Routine.

PRACTICE: Leave the ***PADS-LOGIC*** program, and go to DOS and the LOGFIL Directory CD\PADS\LOGFIL and use DIR. Note the presence of the "CUT" File, with the Name you used and the extension of .GRP. Now, bring back the schematic, and using F2 ADD SHEET, add a new sheet to the schematic. Select F4 GROUP, then F2 PASTE GRP. Respond to the prompt for GROUP FILE NAME with the name given to the "Cut" group. The Cut Group File will be brought from the hard disk and attached to the cursor, ready for you to place and F1 COMPLETE on the new sheet.

Note that the cut connections have been renamed. You must Name the connections that have been cut at the group boundary, as appropriate, to integrate the pasted Group into the design.

6.1.8 GROUP (BUS)

After a bus has had connections attached to it, the moving or rearranging of the Bus MUST use the GROUP features. In this fashion, the bus and associated connections can be Grouped together and moved with a single movement, rather than moving the bus, then moving each of the attached connections.

To move a bus with connections attached, select F4 GROUP and create the Group containing the affected items. After the Bus Group is created, use MOVE, COPY, DELETE, as necessary.

F1 MOVE will relocate the Bus Group to the new location.

F2 COPY will make another Group. Respond to the prompt:

CUT CONNECTIONS AT BOUNDARY (Y/N)?

"Y" will result in the connections within the Bus Group connections being cut and brought with the bus with off-page symbols automatically included.

When a Bus Group is copied:

1) A bus that is completely within the Group will be included in the New Group using the same Bus Name.
2) A bus that is CUT by the Group outline, will have that portion of the Bus within the Group Outline included in the copied Bus Group and will use the Same Bus Name.

In the response to the prompt:

CUT CONNECTIONS AT BOUNDARY (Y/N)?

3) If the response is "Y," the connection within the Group will be copied, an OFF PAGE Symbol will be automatically attached, and the same connection Bit Name will be automatically given to the connection.

If the response is "N", the portion of the Bus within the Group will be copied with the same Bus name, but the cut connections will not

be copied.

4) Parts in the Copied Group will have upwardly incremented REF DES automatically assigned.

PRACTICE: Let's practice BUS GROUP. On Sheet #2 of SELFTRAN, are some Buses with connections. Create a Bus Group that includes the connections from U1A, and the Bus DA [00:11], associated with these connections.

MOVE the New Group. Note the connections continue with the Group in an Orthogonal manner. You may need to tidy up the results.

COPY the Group, responding to the "Cut at Boundary" prompt with "Y". Note the connections have off-page symbols and use the same signal name.

Copy again, but respond "N" to the prompt. Note the connections are not included in the Group.

Repeat the exercise; but this time, Delete the Group and study the results.

6.2 PLOTTING YOUR SCHEMATIC

Let's become acquainted with plotting your schematic. You may plot either a complete sheet, or you may plot only that part of a sheet that is displayed at the CRT. You may plot using Matrix Printers, Pen Plotters, and Photoplotters.

To use the Plotting feature, have your printer/plotter available, the correct plotter driver installed, a schematic in memory, and displayed at the CRT. Select F1 IN/OUT; then F8 PLOT. The Menu changes to:

IN/OUT
PLOT
1 PRINTER
2 PEN-PLOT
3 PHOTO-PLOT

4 APERTURE
0 EXIT

F1 PRINTER, F2 PEN-PLOT and F3 PHOTOPLOT are used to select the Plotting device. F4 APERTURES is used to set up the Apertures for Photoplotting, if the Plotting device is to be a Photoplotter.

After selecting the plotting device, a CRT Screen, as shown in Part III, Chapter 2, Section 2.9.1, is presented. This screen allows you to set up the condition for plotting Sheets, Window, Scale, etc.

Let's do some plotting on your matrix printer. Have the file, DEMO, in memory with Sheet One displayed. Select IN/OUT; then F8 PLOT; then F1 PRINTER.

The screen, as shown in Part III, Chapter 2, will be displayed. We want to plot a Sheet. Enter "N" to the first line. Plot at a scale of 1:1. Select F9 COMPLETE. Assuming the printer driver has been installed correctly, the plot will be made.

Let's do it again, but this time, plot only a portion of the sheet which is displayed at the CRT. Zoom Sheet One of DEMO until only a part of Sheet One is displayed. Repeat the above, but this time, respond "N" to the first and second lines of the Figure in Part III, Chapter 2, Section 2.9.1, and "Y" to the third line indicating Plot Window.

Repeat PLOT, but this time, plot all four sheets of DEMO by responding "Y" to the first Query of the plot screen: PLOT ALL NUMBERED SHEETS.

TUTORIAL SEVEN

THE MACRO ROUTINE

7.1 INTRODUCTION

In this tutorial you will become acquainted with macros, how to create them and how to use them. Macros are a form of shorthand: that is to say, many individual commands can be combined into a single command, thus saving considerable effort on your part.

Part II, Reference Manual, Chapter 13, describes Macros in detail and can be referenced when needed.

The MACRO Management routine is located in the SETUP subroutine as F3 MACROS. When selected, the menu changes to the following:

SETUP
MACROS
1. HELP/START
2. LOAD/FILE
3. SAVE
4. DEL MACRO
0. EXIT

F1 HELP/START, when selected provides a Help file of the Macros using the ALT key.

F2 LOAD FILE is used to bring into memory a file that you have previously created and saved, and intend to use again.

F3 SAVE allows you to save a macro under a macro file name.

F4 DEL MACROS allows you to delete selected or all macros.

7.2 MACRO COMMANDS

There are some basic macro commands that you MUST know. These are as follows:

MACRO COMMANDS

ALT 1 = REPLAYS AUTO MACRO
ALT 2 = CREATE/FINISH AUTO MACRO
ALT 3 = CREATE/FINISH USER MACRO
ALT 4 = SET ABSOLUTE/RELATIVE COORDINATE MODE
ALT 5 =
ALT 6 = SET YES/NO MODE
ALT 7 = RESTORE MENU POSITION
ALT 8 = GO TO TOP MENU
ALT 9 = DISPLAY FILE
ALT 0 = DISPLAY ERRORS
C = REPLAY COUNT FOR USER AND AUTO MACROS

NOTE: ALT 2 and ALT 3 both start AND finish the recording of a Macro.

NOTE: These macro commands are listed under F1 HELP, under MACROS.

7.3 TYPES OF MACROS

There are a number of types of Macros:
1. Auto Macros
2. User Macros
3. Menu Macros
4. Modeless Macros

Types 1, 2, and 3 are true macros and can be created by yourself using the MACRO feature. Type, 4 Modeless Macros are a special form of macros and are provided as part of the ***PADS-LOGIC*** software. They are created by CAD Software, Inc.; facilities are not provided in ***PADS-LOGIC*** to permit the user to create modeless macros.

7.4 CREATING MACROS

You can create a macro using two different techniques:

1. By performing the action at the CRT/keyboard and recording the macro.

2. By using a Text Editor and composing a Macro file in Macro Language.

In this tutorial you will use both methods.

7.5 CREATING AN AUTO MACRO

What is an Auto Macro? An Auto Macro does not take a name and hence only one can be created at any given time. (The other type of macros, the ones with names, are called User Macros). The Auto Macro exists mainly as a quick way to make and execute a temporary macro; this macro cannot be saved to a file.

The other difference between the Auto and User macros is that at the start of Auto Macro creation, the Macro system is automatically placed in Relative coordinate mode.

Auto Macros are created by selecting ALT 2. When the macro is completed, selecting ALT 2 once again, completes the macro. To replay the Auto Macro, select ALT 1.

PRACTICE: We shall now create an Auto Macro.

Creating an Auto Macro

In this exercise we shall create an Auto Macro. What this Auto Macro will be capable of doing is to add a series of 4 gates to the schematic, spaced 1" apart in the X direction.

Here are the steps to accomplish this Auto Macro:

1. Move cursor to coordinate 10000 10000
2. Select ALT 2 (to start recording the macro)
3. Select Add Part F2
4. Type 74LS04 <CR>
5. Select COMPLETE (F1)
6. Move cursor to coordinate 11000 10000
7. Select Exit (F10)
8. Now select ALT 2 (This completes the recording of the Auto Macro)

9. To execute the Auto Macro, type C3 and select ALT 1. (this will allow the macro to be executed 3 times).
10. The macro will be executed 3 times, adding 3 new parts for a total of 4 gates. Notice the Assignment Reference Designators are automatic.

NOTE: While you are recording the Macro, your every action is also recorded. For this reason, we suggest that for the more complex Auto Macros, you write out an outline of what you intend to do before starting the macro.

Try your hand now in making up an Auto Macro to place a number of items (parts, text, 2D lines, etc. on the Y-Axis).

7.6 LOADING USER MACROS

Macro files can be loaded two ways. One way is to put the User macros that you always want loaded in the file, CAE.MAC. This file is located in the \PADS\LOGFIL directory. This file is automatically loaded at program startup.

The second way to load a macro is to load from within Logic by using the LOAD command in the Macro Management menu. The LOAD command will always load all the macros in a file. If any of the macros in the file already exist in memory, the loading process will be aborted.

7.7 CONTROLLING MACRO REPLAY

There are two types of control you have while a macro is replaying; Pausing and Aborting. To pause during replay, hit the space bar. To resume, hit the space bar again. To abort replay, type Ctrl X.

7.8 USER MACROS

User macros are those macros that you expect to repeatedly use over and over (as differentiated from an Auto Macro) and, therefore, after it is created, you save it for future use. The User Macro is always saved in a file called CAE.MAC.

While ALT 2 was the START/STOP command for Auto Macros, the

START/STOP command for User Macros is ALT 3.

7.8.1 WHAT ARE MENU MACROS

MENU MACROS are a special kind of USER MACRO that allows the ***PADS-LOGIC*** User to select a menu item quickly. The User does not need to select menus, and submenus in order to find the executable menu item. By selecting the Menu Macro the ***PADS-LOGIC*** User will be "jumped" to the desired menu item and ready for the next command.

The menu commands are as follows:

ALT E = ECO MENU
ALT X = EXIT TO DOS
ALT M = MACRO MENU
ALT T = TOP OF MENU
ALT L = LIBRARY MENU

The various MENU COMMANDS listed above allow the User to enter the ECO routine, Exit completely to the DOS prompt, get to the MACRO menu item, moves the User to the JOB IN menu, and to the LIBRARY menu.

The COMMANDS listed above are contained in a file called: CAE.MAC. This file is located in the \PADS\LOGFIL directory. If the macros are accidentally deleted, a backup file of CAE.MAC has been included. The file name is CAE.SAV and this file resides in the \PADS\LOGFIL directory. Simply use the DOS copy command: COPY CAE.SAV CAE.MAC.

PRACTICE: Go to various positions of the menu routines and subroutines and then use the different macro commands until you are sufficiently familiar with them, so as to be able to use them to do productive work.

NOTE: If the user has entered **PADS-LOGIC** by utilizing "the Shell," ALT X will exit the user to the Shell. If the user has entered **PADS-LOGIC** by typing "LOGIC," ALT X will exit the user to DOS.

7.8.2 MACRO LANGUAGE

Macros can be created as a text file using a Text Editor and Macro Language. The latter is quite obvious and can be illustrated with some examples.

The following examples are the MENU MACROS you used in the preceding section. They have been created using the PEDIT Text Editor.

Here are the MENU MACROS, and an explanation of each line:

NOTE: The numbers that follow the (f*), are X and Y coordinates, relative to the mouse. These numbers can be omitted.

(bdef) (*ecotest*) (a-e)	Begin definition, macro name, ALT E
(top)	Top Menu
(F1) 3900 12300	In/Out
(F6) 3900 12300	ECO
(edef)	End of definition
(bdef) (*exit*) (a-x)	Begin definition, macro name, ALT X
(yn)y	Set Yes/No mode to Yes
(top)	Top menu
(F10)	Exit
(edef)	End of definition
(bdef) (*macro*) (a-m)	Begin definition, macro name, ALT M
(top)	Top menu
(F8)	Set Up
(F3)	Macros
(edef)	End of definition
(bdef) (*library*) (a-l)	Begin definition, macro name, ALT L
(top)	Top menu
(F9)	Other menu
(F9)	Hier/Lib
(edef)	End of definition
(bdef) (*top*) (a-t)	Begin definition, macro name, ALT T
(top)	Top menu

(F1)	In/Out
(edef)	End of definition

These MENU MACROS are executable from any routine within ***PADS-LOGIC***. The User simply decides which command is needed, and types the ALT character, and the MENU MACRO is executed.

PRACTICE: Using PEDIT or other Text Editors, create a text file for a Menu Macro that will take you to the ADD CONN subroutine from any location in the ***PADS-LOGIC*** menu structure.

7.8.3 USER DESIGN MACROS

User Design Macros are those macros that are of a nature that their use is so frequent during a design that they are saved or stored and then recalled when needed. Generally a User Design Macro is associated with a Design feature such as placing parts, bus connections, etc.

In the following two examples we have included the two User Macros that you may use as a reference. These two User Design Macros are:

The first User Design Macro named ALT F1, can be used to add parts to a schematic sheet. This macro can be used on any schematic, and on any sheet. You will notice that as the parts are added, the Reference Designators are automatic.

The second User Design Macro ALT F2, can be used to add signal lines to a bus. You will notice the signal names are automatically assigned from the name of the bus. If the User has changed the name of the bus, the signal lines to the bus will also change. In this example, the signals will be DA00 through DA13.

First Example: In this exercise you will create a User Macro that can add a series of gates in both X and Y. For this example there will be 3 rows of 4 gates in each row. The gate that we shall use is a 6167 gate.

If you wish to see exactly what you will be creating in this first User

Macro example, select ALT F1. This will replay a User macro created specifically for this lesson that adds all twelve 6167 gates. After you execute the ALT F1 User macro, select the menu item NEW SCHEM to create a new schematic. You may now continue with the lesson and create your own User Macro that will add the 6167 gate to specific X and Y coordinates.

EXAMPLE 1:

Now YOU will create the same macro we called ALT P. This macro will be identical to ALT F1. Select menu item NEW SCH, and erase schematic from the screen. You should be at the top menu which has the menu item (F2) ADD PART.

Search for coordinate 5000 13500 by using the modeless command S5000 13500 <CR>. Alternately use the cursor.

1. Select ALT 3 to start recording the User Macro. Screen prompts:

   WHAT KEY WILL YOU USE TO INVOKE THIS MACRO?

2. Select ALT P
3. If you wish, you may now give a description of the macro using 11 characters or less.
   NOTE: Using a space counts as a character.
4. Select ALT 8 to get to the top menu.
5. Select ALT 4 to toggle between Relative and Absolute mode.
   NOTE: For User Macros, use Relative mode. This allows the mouse to be located at any coordinate.
6. When the mode is set to Relative, the screen prompts:

   MACRO COORDINATE IS NOW RELATIVE

7. Select menu item ADD PART (F2).
8. Name of part type for a new part> Type 6167 <CR>.
9. Select COMPLETE (F1).
10. Note the Reference Designator of the part is U1.
    This assignment is automatic.
11. There is now another component automatically attached to the cursor.

12. Now move the cursor to the coordinate 7000 13500.
13. Select EXIT (F10).
14. Select ALT 3 to stop recording the User Macro.
15. Screen prompts:

MACRO (A-P) CREATED

NOTE: Now that this macro is created, it must be saved to a macro file.

16. To accomplish this, select SETUP (F8).
17. Now select MACROS (F3).
18. Select SAVE (F3).
19. Output filename? (CR=) Type PARTS.MAC <CR>
20. Screen prompts:

MACROS SAVED TO FILE
\PADS\LOGFIL\PARTS.MAC

21. Now select ALT 8, this will bring you to the top menu.

HOW TO REPLAY A USER MACRO

We must now indicate how many times we want this User Macro replayed. In this case the number of replays is 3. This will give us a total of 4 each, 6167 gates in the top row.

The C command specifies the number of times a Macro will be replayed when it is invoked. It is primarily for use with Auto Macros but can be used with User Macros. To replay a macro only once, simply type the Macro name. Ex: ALT P. To replay it more than once, type "C" followed by the number of times you want it to replay, then type the macro name. For instance, to replay the ALT P macro 3 times, type C3 and then ALT P.

We will now replay the User Macro.

1. Move cursor to coordinate 7000 13500 by using the modeless search command, S7000 13500 <CR>.

2. Type C3 (this is a count command that must get typed before

executing the ALT P macro). This flags the macro as to how many times to execute. Now select ALT P; the macro will automatically execute 3 times. Thus completing the top row of gates.

3. Now move the cursor to coordinates 5000 9500 by, once again, using the modeless search command..

4. Type C4 and select ALT P, the macro will execute 4 times. This now completes the second row.

5. Now move the cursor to coordinates 5000 5500, using the modeless search command.

6. Type C4, and select ALT P, the macro will execute 4 times. Thus completing the last row of 6167 gates. You have now laid out 3 rows with 4, 6167 gates in each row.

EXAMPLE 2: ALT F2 (Connections to a Bus)

CAD Software, Inc. has included a User Macro ALT F2. This User macro will demonstrate the process of connecting signal lines to a bus. To execute this macro, select ALT F2. When this macro has executed, select the menu item NEW SCH to create a new schematic. To see this macro, select ALT F2 now.

You may now continue with the lesson and create your own User macro that will add signals to a bus line. We will call this macro ALT S. You will first put in the part and the bus manually. To do this you will first place a part, then create a bus. After this is done, you will create a macro that will make connections between the part and the bus.

a) Using ADD PART, move a 6167 part at location 12000 12000

b) Using ADD BUS, add a bus from location 10000 14000 to location 10000 10000. You may use the modeless search command to find these two coordinates. Type S1000 10000 <CR>.Name the bus DA[00:13]. Center and enlarge the gate and bus line.

When this is done, create a User Macro called ALT S as follows:

CREATING A USER MACRO TO CONNECT SIGNAL LINES TO A BUS

1. This macro will connect the first 19 pins on the Gate.
2. Place cursor over pin 1 of U1. The coordinate is 12000 10300.

NOTE: The number lock key must be turned off, in order for the numeric keys to work.

3. Select ALT 3, to start recording the User Macro.
4. Screen prompts:

   What key will you use to invoke this macro:

5. Select ALT S keys.
6. Macro description (11 chars max, (CR) for none) Type:

   SIGNALS <CR>

7. Screen prompts:

   NOW CREATING MACRO (A-S)

8. Select ALT 8 for Top menu.
9. Select ALT 4 for Relative mode.

NOTE: If screen prompts Absolute mode, select ALT 4 once more to toggle to relative mode.

10. Screen prompts:

    MACRO COORDINATE MODE IS NOW RELATIVE

11. Place cursor over pin 1 of U1, and select ADD CONN (F3). The coordinate is 12000 10300.
12. Move cursor to bus line.
13. Select CRN/FINISH (F1).
14. Net Name > DA00 prompts at the bottom of the screen.
15. Select MOVE (F2).
16. Move signal name to appropriate location and select COMPLETE (F1).

17. Once again, select COMPLETE (F1).
18. Place cursor over Pin 2, to give correct spacing of signal lines. The coordinate is 12000 10400.

NOTE: This coordinate is locating pin 2 of U1.

19. Select ALT 3.
20. Screen prompts:

MACRO (A-S) CREATED

NOTE: Now this User Macro must be written to a macro file.

21. Select SETUP (F8).
22. Select MACROS (F3).
23. Select SAVE (F3).
24. Output filename? (CR=\PADS\LOGFIL\PART.MAC) Type:

SIGNALS.MAC <CR>

25. Screen prompts:

MACROS SAVED TO FILE
\PADS\LOGFIL\SIGNALS.MAC)

26. Select EXIT (F10).
27. Once again, select EXIT (F10).

TO REPLAY THIS USER MACRO

1. Make sure the cursor is placed over pin 2 of U1. The coordinate is 12000 10400.

2. Type C13 and select ALT S. This will add all the signals to the pins on the part.

NOTE 1: Notice the signal names appear in the proper location. This occurs because of the location of the first signal name which you placed.

NOTE 2: If a User Macro which contains signals that will connect

to a bus line executes, but does not add the connections, the following is an explanation and instructions for correctly creating the Macro:

User Macros are very sensitive to cursor coordinates. At the time of creating the Macro, the cursor should be at the proper coordinate, in this example, over pin 1 of U1 (the coordinate is 12000 10300). When the signal is connected to the bus line, the cursor should be placed on pin 2 of U1 at the coordinate 12000 10400. If the cursor coordinates are not observed when creating the Macro, the cursor will not find the object. Thus, the macro will not execute correctly.

TUTORIAL EIGHT

USING THE HIERARCHICAL ROUTINE

In this Tutorial we will cover all aspects of the Hierarchical Routine and in the process, create a drawing containing Three Hierarchical levels.

In the process of performing this Tutorial, it would be best if you follow our instructions precisely. You will be covering a fairly complex routine and we hope to instruct you in how to use the routine and, in the process, have you construct a multi-level Hierarchical design.

The Hierarchical Routine is located under the OTHER MENU Routine and is identified as F9 HIER/LIB. When selected, HIER is identified as F7 HIERARCHY.

At this point, for background purposes, please go to Part III, the Reference Manual, Chapter 12, and read 12.0 Hierarchical Background. This will provide an overview of the Hierarchical concept. Following this, go to the end of this same chapter and read the NOTES on the Hierarchical Routine in Section 12.5. With this background reading material behind you, you are ready to start your Hierarchical Design. The design that you will create is shown in Figure 8.1A through 8.1I. In the short section which follows, we will describe the contents of each of these sheets so as to give you an overview of what you will be doing.

NOTE: The schematic depicted is a totally nonfunctional design; instead, it is simply a convenient collection of various parts and connections made up to illustrate the concept of different levels of Hierarchy.

Figure 8.1A - This is Sheet #1 of a design and, of course, is at Level 0. Note that it contains a mixture of circuits and Hierarchical Symbols, i.e., Hierarchical Symbol AAA1 and BBB1. Note we suggest the last character of the Symbol Name be used to indicate the level of these Hierarchical Symbols.

Figure 8.1B - This is Sheet #2 of the schematic. Note that it also contains circuit contents and a Level 1 Hierarchical Symbol. One could have just as easily placed all three Hierarchical Symbols on Sheet #1. Naturally, the circuit contents of this sheet are at Level 0.

Figure 8.1C - This sheet is a Hierarchical Sheet (not a sheet in the Design Sequence) and contains the circuitry for Hierarchical Symbol CCC1 and, in addition contains a Hierarchical Symbol that will be at Level 2, i.e., DDD2.

Figure 8.1D - This contains the circuitry for Hierarchical Symbol DDD2 and consists of a small amount of circuitry as well as three Hierarchical Symbols that are at Level 3, i.e., EEE3, FFF3 and GGG3. Note, however, that the contents of these last three symbols, instead of being on a Hierarchical Sheet, will be on a sheet within the design, thus necessitating five sheets to the design, instead of the two original sheets.

Figures 8.1E,F&G - These are the circuit contents of EEE3, FFF3 and GGG3. Note that they are on a sheet within the Numbered Sheet Sequence of the design, rather than a Hierarchical Sheet.

Figure 8.1H - This is the circuit contents of Hierarchical Symbol AAA1. Note that this is on a Hierarchical Sheet rather than a Sheet within a design and it is at Level 1.

Figure 8.1I - This contains the circuit contents of Hierarchical Symbol BBB1 and is also a Hierarchical Sheet.

You are now going to create these various sheets of the drawing and Hierarchical Symbols, following which you will plot the sheets, following which you will generate a Net List of the entire design, and finally, you will study the plots as well as the Net List to obtain a good appreciation between the various levels of the Hierarchical design and how Connection Nets and Reference Designations are handled in a Hierarchical Design.

We will organize the creation of this design in structured steps and we would recommend that when you are doing a Hierarchical Design, you should follow a similarly structured series of steps.

1. In a real life situation, we would suggest that before you actually start creating the design that you rough draft the contents of each Sheet, and name the Hierarchical Symbols, making a decision as to whether they will be Hierarchical Sheets or Design Sheets and paying particular attention to those connection nets that will travel across Hierarchical Levels.

2. Let's start the design. In order to expedite the entire process, especially the plotting and Net List, let's use a small A-size sheet. Go to SETUP, PARAMETERS and change the Sheet Size from D Size to A Size. Note that at the CRT you now have an A-Size sheet.

3. Using IN/OUT and F5 SHEETS, make the drawing a two-sheet drawing, using ADD SHEET. (The Drawing will eventually become a Five Numbered Sheet Drawing, since we will later make some Sheets as Numbered Sheets, rather than Hierarchical Sheets.) Now return to Sheet #1.

4. Let's identify the sheets now: In the title block on Sheet #1, put in the Name of the Drawing, HIER, and then Sheet #1, using OTHER MENU and ADD TEXT. Change the Text size to 200 Mils High with 10 Mils Line Width. Do the same thing to identify Sheet #2.

5. Go back to Sheet #1 and create the circuitry including the IC's as shown in Figure 8.1A, the Bus and the Connections. Pay particular attention to naming the Off-page sheets. The Part Type name of P1 is CON\26P\ED.

6. Now that the circuitry that is to be contained on Sheet #1 is completed, create the Hierarchical Symbols of AAA1 and BBB1. Before doing this, however, go to Chapter 15 of Part III, the Reference Manual, and read Section 15.1 on how to Create a New Hierarchical Component.

7. In creating Hierarchical Symbol AAA1, respond Yes to the prompt:

CREATE NEW HIERARCHICAL SHEET

since you have decided both Symbols AAA1 and BBB1 will be on

Hierarchical Sheets rather than sheets of the Drawing.

Referring to Figure 8.1A for Symbol AAA1, you will note there is an Input and an Output pin. Name them A and B respectively, using F3 TERMINALS and then F5 SET PNAME. Notice that the Automatic incrementing module allows you to enter A for the first pin and then simply point at the second pin and it is named B. Finish the completion of Hierarchical Symbol AAA1 with F9 COMPLETE and then place it on Sheet #1 as appropriate and finish with F1 COMPLETE. Repeat the same procedure for Symbol BBB1.

8. You have completed Sheet #1. Make a Job Save of the design using the JOB OUT subroutine and the file name HIER.

9. We now want to create the circuit content for the Hierarchical Symbol AAA1. Go to the Hierarchy Routine and either select F8 HIER IN or the Modeless Command HI. Either of these commands will bring up the Hierarchical Sheet Outline. This Outline is in White to indicate to you that it is a Hierarchical Sheet rather than a numbered sheet in the design. Expand the White Outline to a scale to enable you to do work on it and Exit back to the top menu. Using Figure 8.1H as a guide, create the circuitry for Hierarchical Symbol AAA1.

Upon completion of the circuit, including the naming of the Inputs and Outputs, Exit from the circuit by using F9 HIER OUT. Repeat the same sequence for the creation of the circuit for Hierarchical Symbol BBB1, using Figure 8.1I as your guide. When completed, Exit from the circuit using either F9 HIER OUT or the Modeless Command HO.

10. This completes Sheet #1, including Hierarchical Symbols and their circuitry. Save the job now using JOB OUT.

11. Go to Sheet #2 and create the circuitry found on Sheet #2 (Figure 8.1B).

12. When completed with the circuitry, create the Hierarchical Symbol CCC1 using a Hierarchical Sheet (Figure 8.1C). When complete, place the CCC1 Hierarchical Symbol on Sheet #2 as

appropriate.

13. Create the internal circuitry for CCC1, using Figure 8.1C as a guide. Use F8 HIER IN to bring up the Outline of the Hierarchical Sheet. Expand it and create the circuitry for CCC1.

14. Note that in addition to the circuitry on CCC1, there is also a Hierarchical Symbol, DDD2. This, of course, will be at Level 2. At this point, create the Hierarchical Symbol for DDD2, using a Hierarchical Sheet. Note that there is one Input and two Outputs. When complete, place on the Hierarchical Sheet as appropriate and Exit with either F9 HIER OUT or the Modeless Command HO.

15. You should be back on Sheet #2 (Figure 8.1B) now. Make a Job Save using JOB OUT.

NOTE: The reason that we make Job Saves periodically is in the event that you have a system/power failute, you will not have to start from the beginning, but rather can call up the latest file and start from this. (Alternatively, you could call up the Backup file.)

16. You should now have Sheet #2 displayed at the CRT. You will recall that we have not yet created the circuitry for the Hierarchical Symbol DDD2 that was contained with the Hierarchical Symbol CCC1 on Sheet 2. Let's do that at this point. Go to HIERARCHY and use F8 HIER IN or the Modeless Command HI to go down to the level wherein we can create the circuitry for DDD2. Select F8 HIER IN, then use the cursor to select the Hierarchical Symbol and the contents of CCC1 will be displayed. Repeat F8 HIER IN on DDD2 and a blank Hierarchical Sheet Outline is displayed. Use Figure 8.1D as a guide to create the contents of DDD2. First create the circuitry; following this, create the Hierarchical Symbol for EEE3. This time, however, rather than using a Hierarchical Sheet, we will use a numbered sheet within the numbered sequence of the drawing. Therefore, respond No to the prompt:

CREATE NEW HIERARCHICAL SHEET

In response to the Sheet Number, respond 3. You will receive a message that Sheet #3 does not exist. This is because you started off creating a two-sheet drawing, therefore, you will have to Exit

from HIERARCHY, go to INPUT/OUTPUT and create a Sheet #3. When Sheet #3 is completed, return to HIERARCHY and complete the Symbol for EEE3.

17. Note that when the Symbol is completed and displayed, (1) it also contains the sheet number in the design, i.e., Sheet #3, and (2) the Input/Output pins are not displayed. With regard to the latter, this means that when you are creating the circuit contents for EEE3, you will have to pay particular attention to Signal Names that are to be tied in with other Hierarchical Levels as well as on the Level 0 sheets of the design. When complete, place the Hierarchical Symbol.

18. Create the Hierarchical Symbol for FFF3. On FFF3, also use a Sheet of the Design rather than a Hierarchical Sheet. When asked the Sheet Number, respond 4 for Sheet #4. Again, note that you received a message:

SHEET NUMBER DOES NOT EXIST

You will need to return to IN/OUT and SHEETS and create Sheet #4. While you are there, create Sheet #5 for use with the Hierarchical Symbol GGG3. Return to HIERARCHY now and create the symbol for Hierarchical Symbol FFF3, using Sheet #4.

19. We now create the circuit contents of Symbols EEE3, FFF3 and GGG3. You will note from Figures 8.1E,F&G that the contents of all three Hierarchical Symbols are the same (except for different Reference Designation numbers, of course). Here is an easy way to create the contents of all three symbols.

20. Go to Sheet #3 and create the circuit contents as shown in Figure 8.1E Once created, use the routine GROUP to cut and save the file to hard disk. Now move to Sheet #4 and paste this cut file on Sheet #4. Notice that the Reference Designations have been automatically incremented for you. Now go to Sheet #5 and paste the cut group on Sheet #5.

This completes the "generating" portion of this Hierarchical Design. It is always wise to examine the contents of Hierarchical Design immediately after creating. To do this, we will first plot all the

sheets and then create a Net List and examine.

Go to INPUT/OUTPUT, then plot. In response to the question:

PLOT ALL NUMBERED SHEETS (Y/N)

Respond Y since you want all numbered sheets at this time to be plotted. On the second line, which reads:

PLOT ALL HIERARCHICAL SHEETS (Y/N)

also respond Y, since you want the Hierarchical Sheets plotted. Then proceed to performing the plot using F9 COMPLETE. All sheets of the design will now be plotted with the Numbered sheets being plotted first, followed by the Hierarchical Sheets.

Next, we want a Net List of the Design, therefore, go to REPORTS, then NET LIST. Select ***PADS-PCB*** or any of the board CAD Input programs, and generate a Net List Output file. Then print this file. Now compare the Net List against the plotted sheets of the design. If you have proceeded correctly in using the Hierarchical features, you will note that the Nets A, B, and 3, which traverse essentially all Levels of the design, have each had their nodes combined as appropriate. You will also notice that the Reference Designations of all of the components have been automatically incremented correctly.

This Tutorial has <u>not</u> been a simple exercise and you are to be complemented if you have completed the design correctly and have a good grasp of the Hierarchical concept.

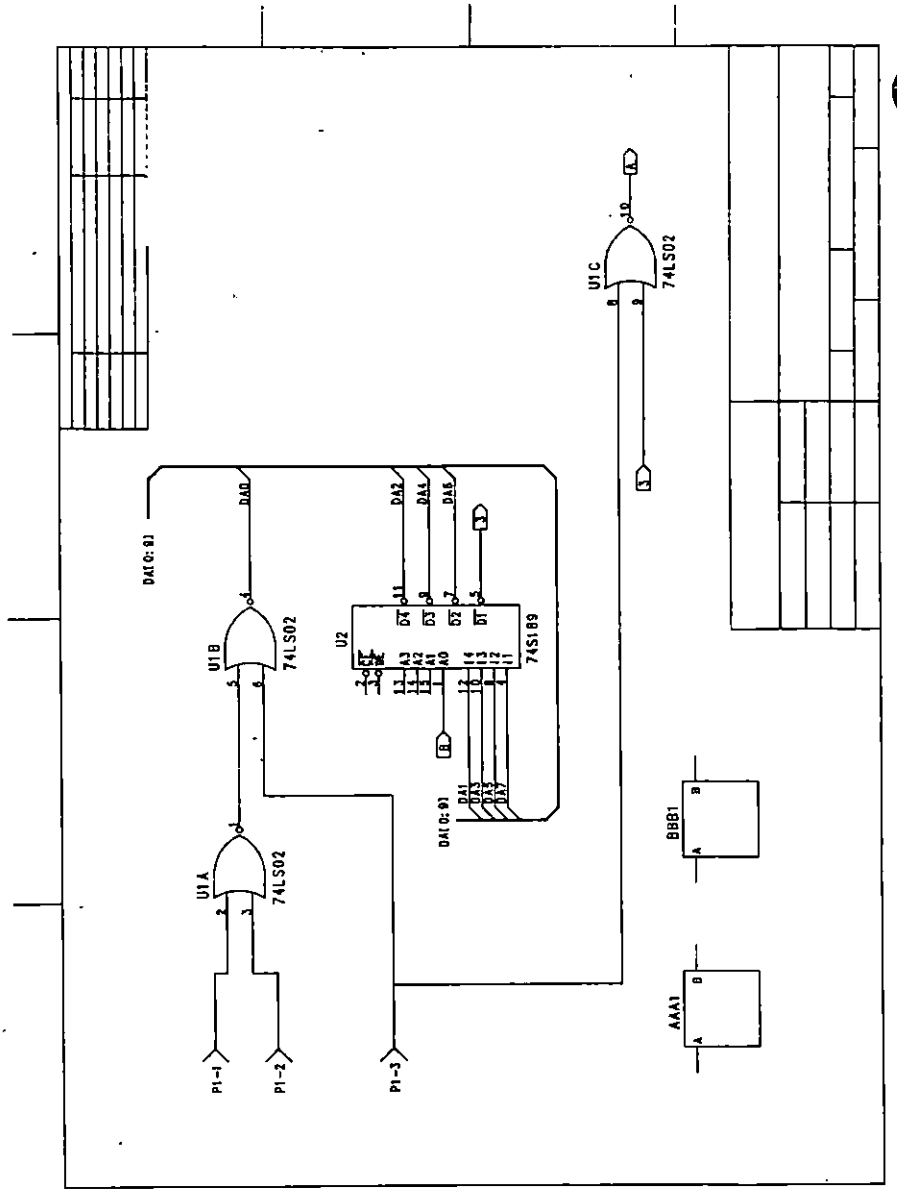


Figure 8.1A - DESIGN SHEET 1 (LEVEL 1)

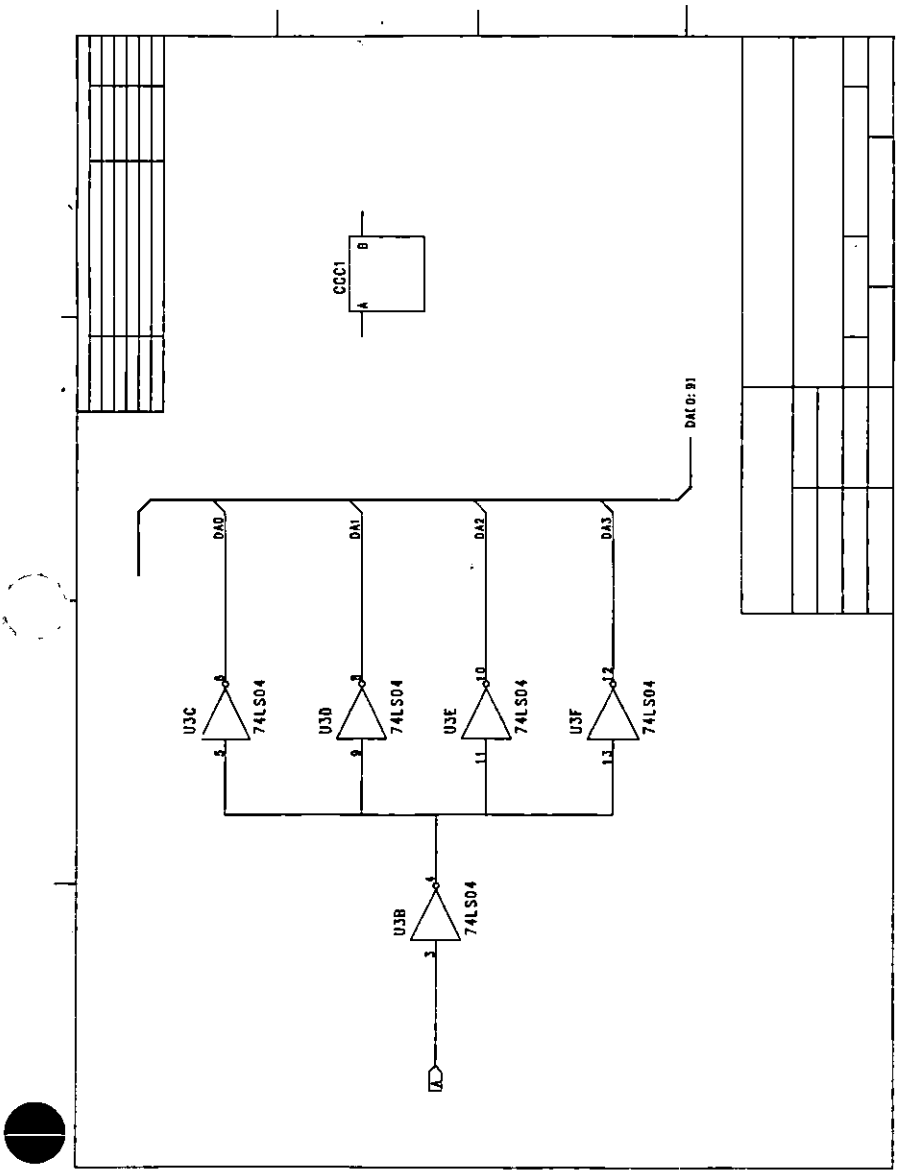


Figure 8.1B - DESIGN SHEET 2 (LEVEL 2)

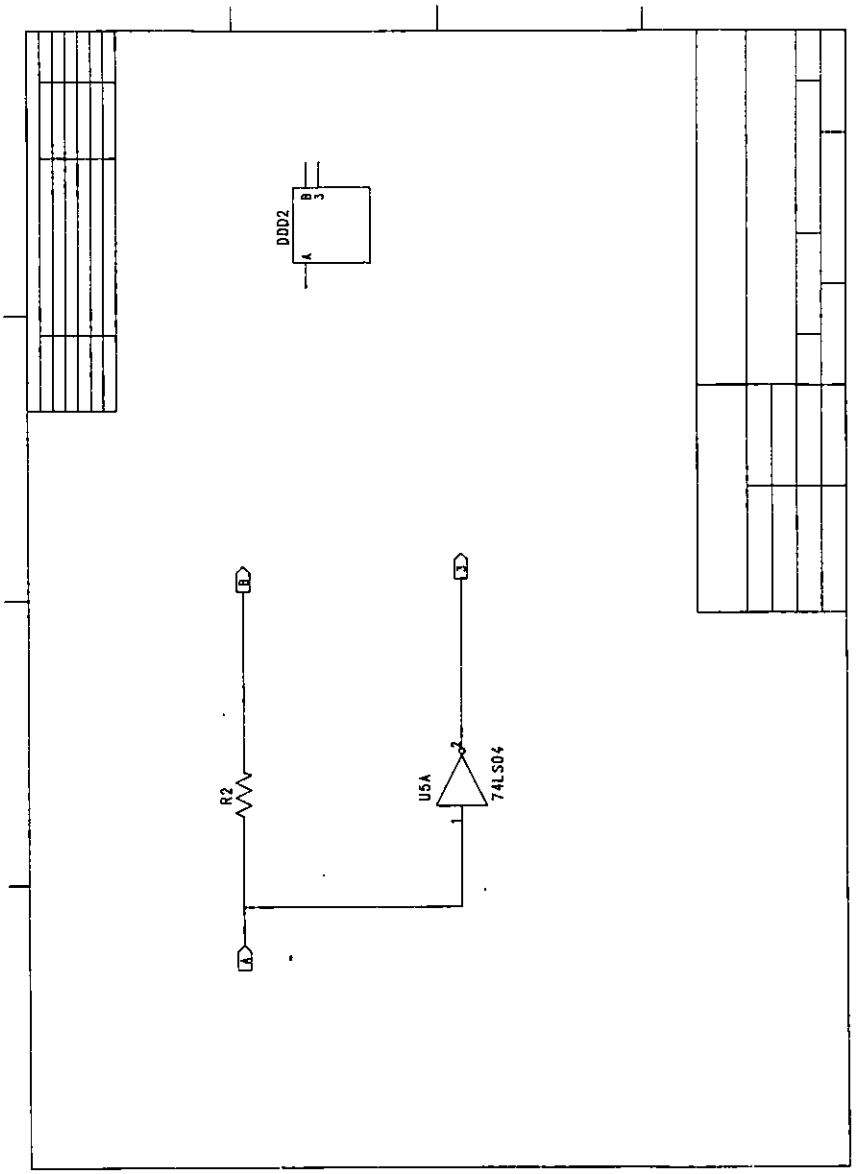


Figure 8.1C
CIRCUITRY FOR HEIRARCHICAL SYMBOL CCC1 - LEVEL 1

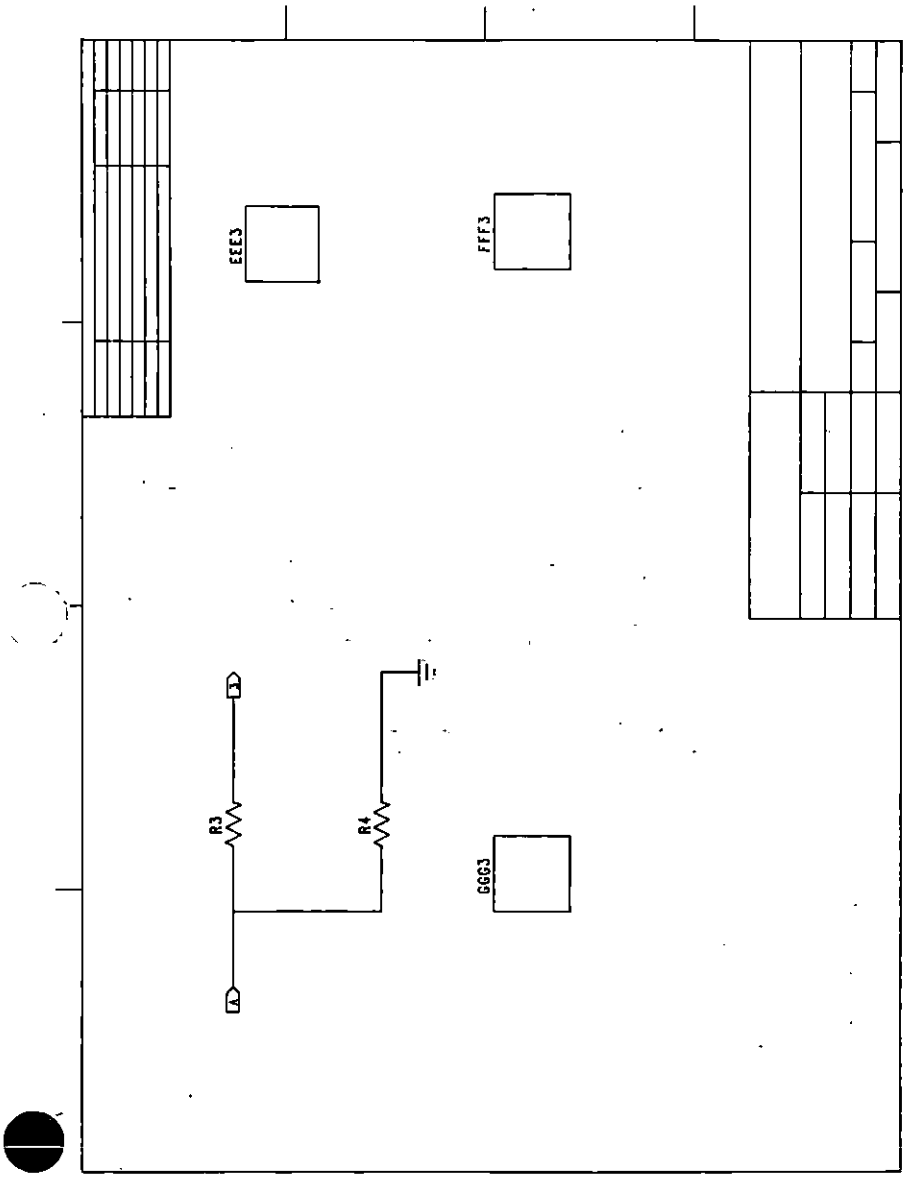


Fig 8.1D Circuitry of Symbol DDD2

Figure 8.1D
CIRCUITRY OF HIERARCHICAL SYMBOL DDD2 - LEVEL 2

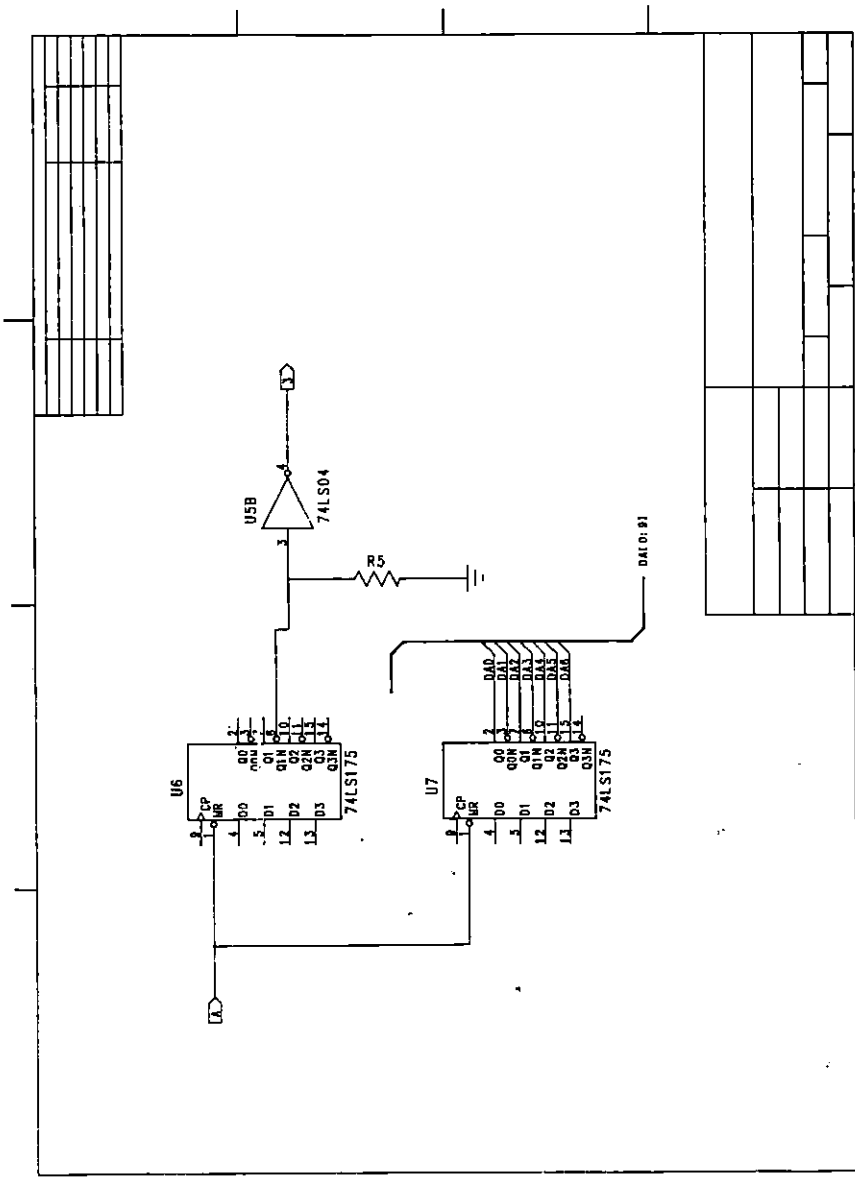


Fig 8.1E uitry of EEE3

Figure 8.1E - SHEET 3
CIRCUITRY OF HIERARCHICAL SYMBOL EEE3 - LEVEL 3

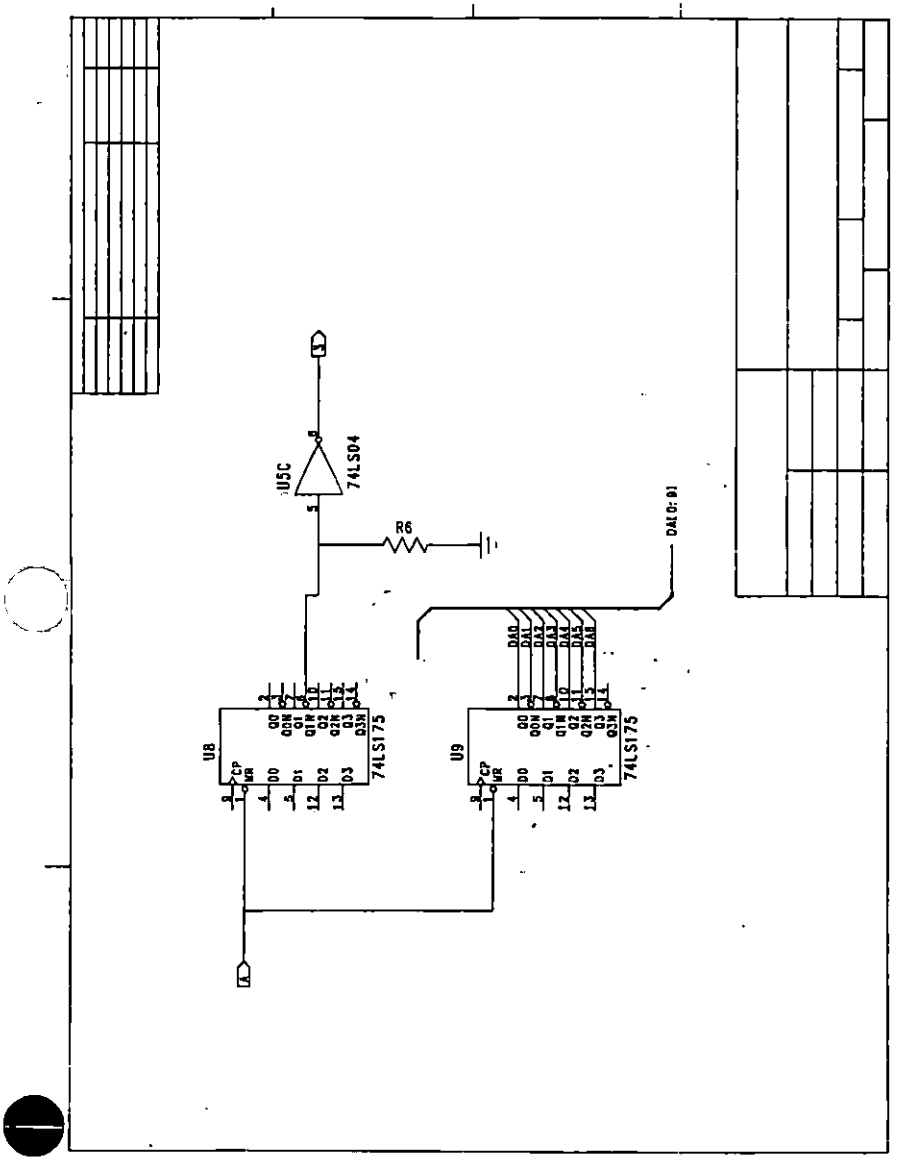


Fig 8.1F Circuitry of FFF3

Figure 8.1F - SHEET 4
CIRCUITRY OF HIERARCHICAL SYMBOL FFF3 - LEVEL 3

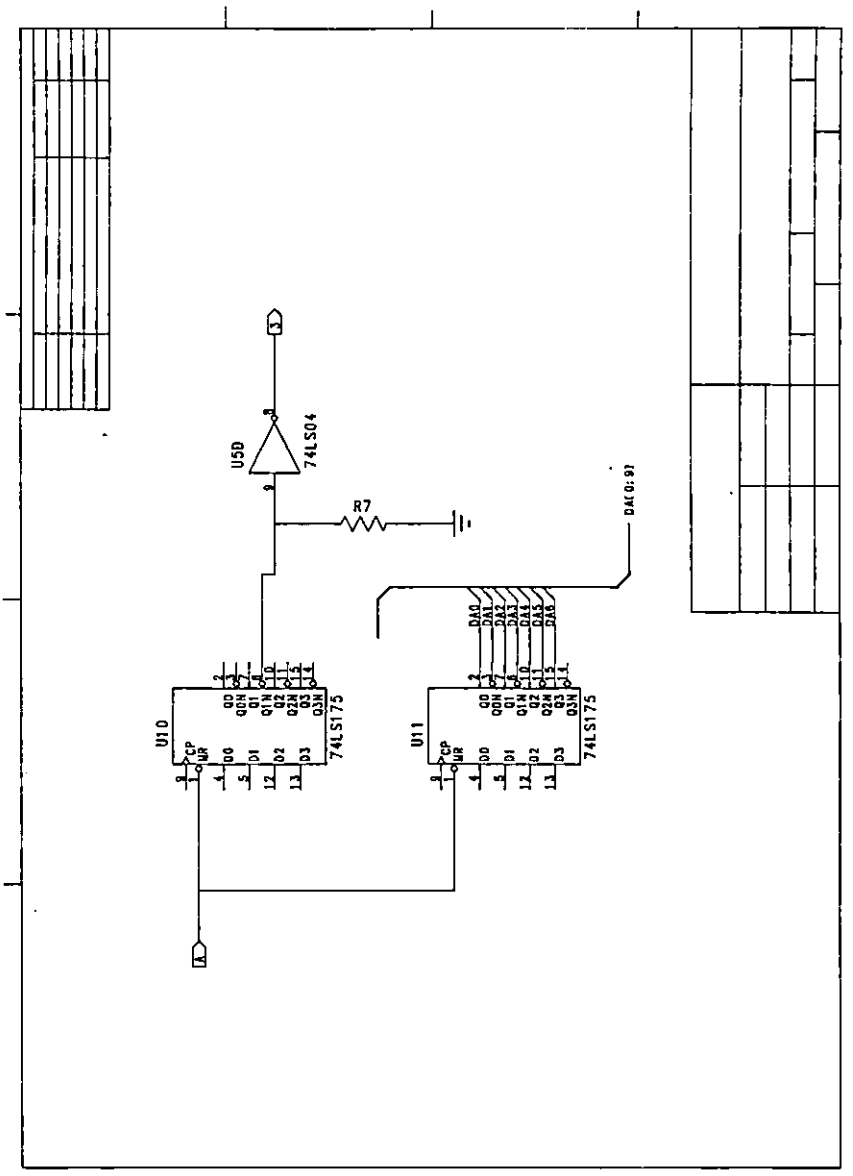


Fig 8.1G Circu of Symbol GGG3

Figure 8.1G - SHEET 5
CIRCUITRY OF HIERARCHICAL SYMBOL GGG3 - LEVEL 3

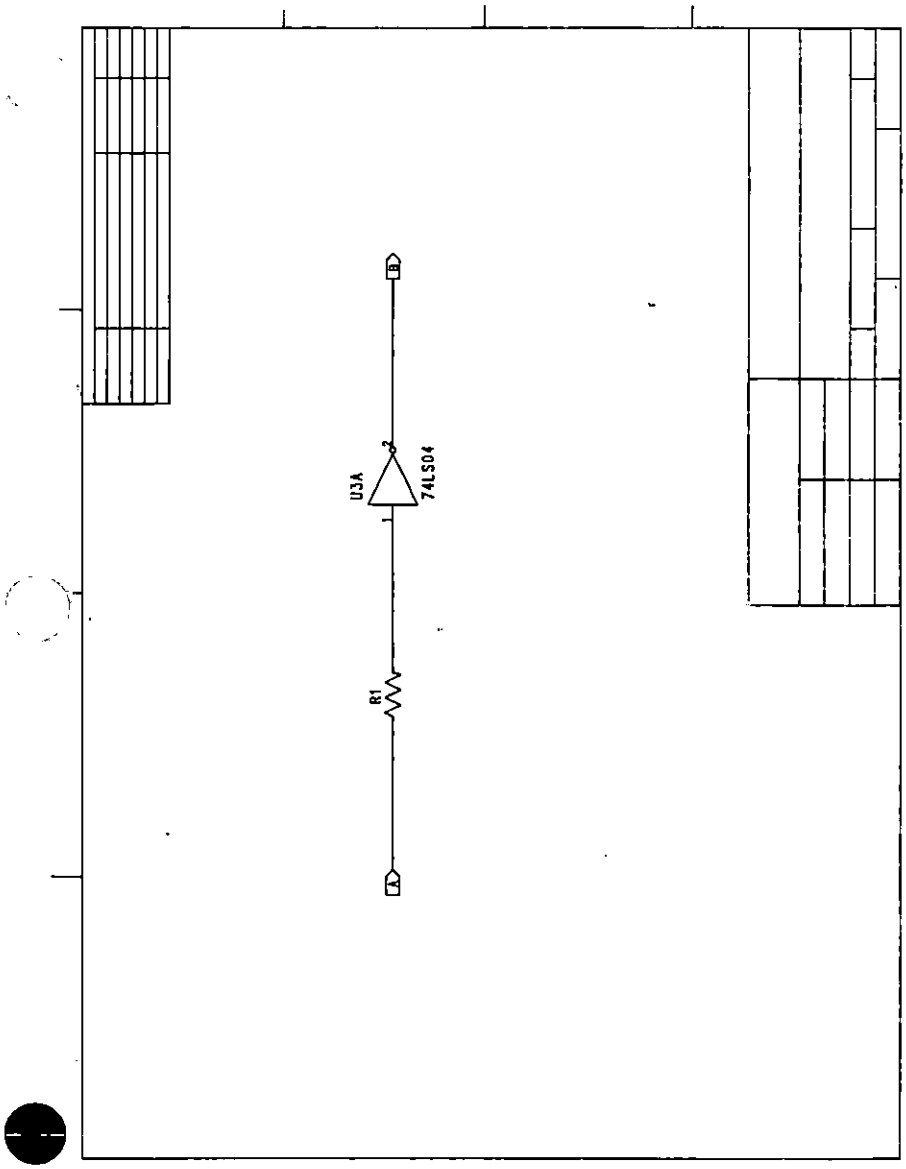


Figure 8.1H
CIRCUITRY OF HIERARCHICAL SYMBOL AAA1, LEVEL 1

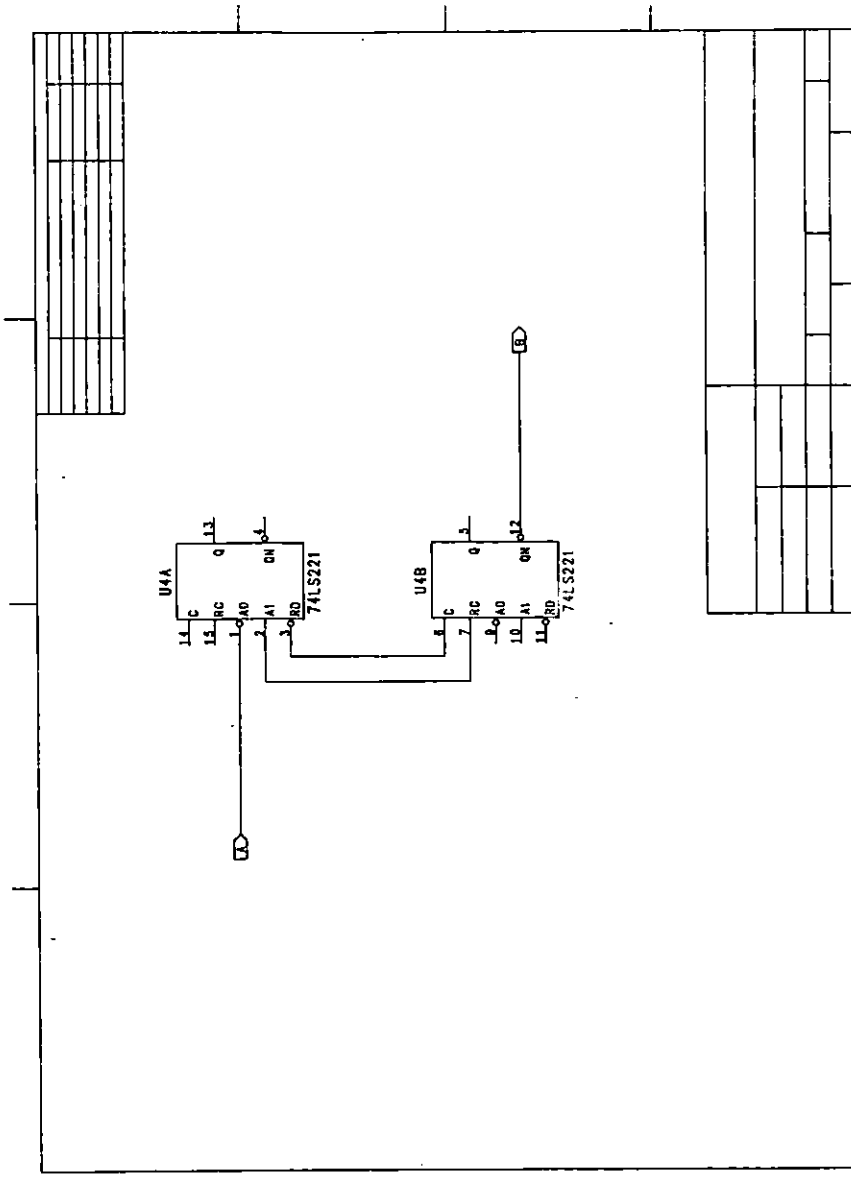


Figure 8.1I
CIRCUITRY OF HIERARCHICAL SYMBOL BBB1, LEVEL 1

TUTORIAL NINE

USING THE THE LIBRARY ROUTINE

In this tutorial, you will learn how to create new Library parts, including the Electrical information as well as the Decal information. Then you will learn how to modify parts that you have created or parts that CAD Software, Inc. has provided in the Library.

In this tutorial, the following items will be created: (1) the complete Part and Decal for a 74LS02 (2) a single Gate item (a Resistor), (3) a ten-pin connector, and (4) a new Decal.

The LIBRARY Subroutine is located in the Top Menu of OTHER MENU as F9 HIER/LIB. When F9 HIER/LIB is selected, the following menu appears:

OTHER MENU
HIER/LIB
1 PART TYPE
2 CONNECTOR
3 OFF PAGE
4 REP OFF PAGE
5 GATE DECAL
6 PIN DECAL
7 HIERARCHY
8
9
0 EXIT

9.1 CREATION OF A MULTI-GATED (74LS02 IC) COMPONENT

In this section of the Tutorial, you will create a TTL Integrated Circuit. To give you a preview of what you will be creating, go to the LIB routine and select F1 PART TYPE. In response to the request for the Part Type, respond 74LS02. The part will be recalled from the Library and the Gate portion will be displayed at the CRT.

Note that there are two parts to making a Part. The first is Electrical

information and the second is the Gate portion which consists of the Decal used to display the part at the CRT, together with Pinning information. You will note these under the PART LIB menu as:

1 ELECTRICAL
2 MOD GATE

You know from your experience that a 74LS02 is a 2-Input Positive NOR device containing four Gates. When looking at the Gate information at the CRT, you see eight gates. The reason for this is that the Primary Gating information is shown in the four gates on the left, and the DeMorgan Equivalent Gates information is shown on the right. Pan around over this information, paying particular attention to the Decals, the Pin Number and the Pin Type and Swap information associated with the Pin. Study this information carefully since you will be creating the information on your own later on.

Next, select F1 ELECTRICAL and the Electrical information for the Part will be displayed.

Study this information while reading Part III, Section 11.1, PART TYPE. Study the entries in the Electrical section for a 74LS02 and based on the description for the Electrical information in Chapter 11, try to determine why the entries are as shown.

When you are satisfied that you have a reasonable understanding of the contents of the Electrical section of a Part, EXIT and bring up the PART TYPE routine. In response to the Name of the New Part, use the name \022 rather than \02. The reason for using this name is to insure that you do not overwrite the "good contents" of the 74LS02 as contained in the Library.

What you will actually be making is a part we will call 74LS022--a non-existent part but based on a 74LS02--that you can delete from the Library after you have learned the fundamentals of creating an IC part.

The first question that we will address is what to call the Part. We could indeed call it simply 74LS022. That is perfectly legitimate and if it were one of a kind, rather than a member of a family, that's

what you would call it. You will recall that the 74LS02 is a member of a large family, 74LS02, 54LS02, etc., and we do not want to create a part for each of these members of the family when making our \022 part, and thus we use the concept of Suffixes and Prefixes. If you don't recall the exact definition of Suffix and Prefix, go to Part III, Chapter 11, Section 11.1 under F1 ELECTRICAL, and read the paragraphs on Suffix and Prefix.

After having read this section, you recognize that we will call the Suffix of the Part \022 and thus we respond to the Part Type Name as \022.

The system informs you that \022 does not exist, therefore, respond to the prompt that you wish to create the part.

Assuming a NOR Decal exists (it does), we will want to complete the Electrical data first, therefore, when the new menu is presented, select F1 ELECTRICAL and the Electrical Text Screen will be presented at the CRT. Notice that the Suffix has already been entered as a \022.

Prefix
Here is where we identify the entire family for the 022. Go to your IC Data Book for a 7402 and determine the number of different 022's supplied by the vendor and enter the Prefixes in accordance with the instructions for Prefixes in Chapter 11, Section 11.2. Upon completion, your Prefix entry should look approximately as follows:

X4 X4ALS X4LS X4S 54L X4HCT

Logic Family
Since this is a TTL Part, enter TTL. The significance of this entry amongst other things is that in accordance with the data contained in the PKGTYP.DAT file, anytime you use a TTL part, the Reference Designation will start with a U. (Later on, we will show you how to read and modify the PKGTYP.DAT file.)

PCB Decal and Alternates
This is a 14-pin DIP Part and the ***PADS-PCB*** Decal is called DIP14 therefore, enter DIP14.

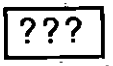
NOTE: In the event that your Printed Circuit Board CAD System is not a ***PADS-PCB*** system, enter the Board Decal Name that you use in your board CAD system for a 74LS02.

Number of Part Attributes
Let's enter the maximum amount of data, i.e., ten lines of data, so therefore enter 10, and ten lines for Attribute data entry will be displayed.

On the first line, enter the Parts Description.

On the second line, enter the Unit Cost.

The third Attribute Line is for Part Type for Digital parts and VALUE for Discretes. If the Part is a Discrete, i.e., Resistor, enter VALUE: ??? If the third Attribute Line is selected for display at the CRT, a Reserved Text Box:



appears with the part when brought on to a sheet of Schematic. When you are creating a sheet of the schematic, use EDIT TEXT to change the "???" to the value of the Part, i.e., 10K Ohms, etc. If an IC, enter the Part Type Number, i.e., 74LS022.

The fourth and fifth Attribute Lines are used to indicate the Manufacturer. Alternately, on a Discrete Part the fifth line can be used to indicate Tolerance, i.e., 1%, 5%, etc. To use, enter TOLERANCE: ??? When the part is brought on to the schematic, EDIT TEXT can be used to change the "???" to correct the Tolerance.

NOTE: All lines must start with a description, followed by a (:). It is the characters following the colon that can be displayed.

NOTE: You may use any number of Attributes from 0 to 10.

Displayed Attribute Line
Let's assume that we wanted to display and plot the Part Description and Part Number. Enter in the first blank, the number "1" to Display/Plot the First Attribute Line of data, and in the second

blank, enter "3" to indicate the third line of data or Part Type is to be displayed at the CRT and plotted on the plotter.

Non-Numeric Pin Number
14-Pin DIPS use Numeric Pin Numbers, therefore, leave this entry as N (No) for the Default.

Number of Signal Pins
The 74LS022 has two Standard signal pins, i.e., GND and +5V, therefore, enter 2.

In the first data entry line, enter information for Ground, which would read Pin Number 7, Signal Name is GND and 50 Mil Track Width. On the second line, enter Pin Number 14, Signal Name is +5V and 50 Mil Track Width.

By entering the "standard" Power and Ground Pin data into the IC Library Part, the Power and Ground Connection to the Net for this part is automatically created when you add the part to the Schematic. Connecting the Power and Ground Pins of the part to a Power and Ground Symbol is not necessary!

Number of Gates
In the data entry lines of Gate data, enter 4 and four lines for Gate data will appear.

The Gate Type refers to the Swappability of each gate within the same IC or with gates of the same part on the board. A 0 indicates "No Swapping Allowed;" a 1 indicates the Gate may be swapped with the same package or other packages of the same Part Type.

The Decal for each Gate will be a NOR Gate, therefore, enter NOR2, and since there is a DeMorgan Equivalent, provide this Decal information also, by entering the DeMorgan Equivalent Decal Name which is NORDM, with the Primary Gate Decal and the DeMorgan Equivalent separated by a space. Enter this information for all four gates. When finished with the Electrical information, ALWAYS use F9 COMPLETE.

NOTE: The Warning that appears at this time is telling you the Part Gate information has not been finished.

Next, you will complete the Gate information. Select F2 MOD GATE.

Notice that the system has automatically created the four Primary Gates on the left hand side and the four DeMorgan Equivalent Gates on the right hand side of the CRT. If you examine these Gates carefully, you will notice that the Pin Numbers are absent, as are the Pin Type and Swappability information. What you must do at this time is to complete this information by modifying each gate.

In response to the prompt:

ENTER GATE NUMBER TO EDIT

enter Gate. A pop-up window appears allowing you a choice of gates; pick NOR2. Gate 1 will be displayed. Using TERMINALS and F4 SET PIN NUMBER, assign the proper pin numbers to Gate 1. Notice the Numbers Autoincrement with each selection of the Left mouse button.

Using F6 SET PIN TYPE, assign the correct Pin Type (Inputs, Outputs, etc.)

Using F7 SET PSWAP, set Swappability information. The two Inputs are Swappable (1); the Output is Not Swappable (0).

Upon completion, carefully examine the Gate and Exit with F9 COMPLETE.

You will notice that the Gate information for the four Gates are redisplayed with Gate 1 completed as per your recent edits. Perform the same operation on Gates 2, 3 and 4 (B, C and D).

NOTE 1: Notice that the Pin Type and Swappability have been automatically duplicated on the three remaining gates as well as the DeMorgan Equivalent, thus saving you considerable time!

NOTE 2: Notice that, in addition, the Pin information assigned to the Primary Gate has been automatically duplicated on the DeMorgan Equivalent.

Upon the completion of both the Electrical and Gate information, you should run a Check on your Part, using F8 CHECK PART. Correct any error messages before you save the Part. Upon receipt of a "No Errors" message, use F9 SAVE PART. It will, at this point, prompt you with the Library and Part Type Name it has been assigned. If all is well, proceed with a <CR>. Watch for the "PART SAVED" message. If in doubt, save again!

NOTE: We recommend that in view of the time you have invested in making a Part, that before you actually need to use it as part of the original creation of the Part, you call it up on a blank Schematic sheet, using ADD PART, and construct a small schematic using the Part; then make a Net List Out as well as run any other REPORTS necessary. This insures that when you are ready to use the new Part, it indeed will be ready to use.

When you are satisfied that you understand how to create multi-gated IC's, delete this \022 Part from your library, inasmuch as it is a non-part. (Use PLIB to delete the Part--see Part III, Chapter 11, Section 11.2 PLIB, on the use of PLIB.

NOTE: In order for the MOD GATE routine to display the Gates as you have seen, the Gate Decal must have already been created. As a guideline, therefore, before starting to create a new part, always insure that the Gate Decal exists in the library.

9.2 CREATION OF A DISCRETE PART

In this section of the tutorial, we will create a simple, single-gate (single Decal) component, specifically, a Resistor.

In general, the creation of a Discrete Part is identical to a multi-gated IC except for certain specifics which will be brought out as you create the Resistor.

Go to HIER/LIB, and using F1 PART TYPE, call up the named R1/8W and examine the ELECTRICAL and GATE information for a Discrete Part.

We will call the new Resistor we are going to make, R1/4WW to distinguish it from the real R1/4W Part that is in the Library.

Before we start making the Part, we want to insure that a correct Gate Decal exists; if not, we will need to create one.

Find out whether a Gate Decal exists. Go to the routine F5 GATE DECAL and in response to the request for the Name, enter RES*. We are making an educated guess that the Decal for a Resistor is called Resistor or something similar and, therefore, by using RES with the wild card *, we are asking for a listing of all Gate Decals starting with RES to be displayed at the CRT. For your information, the Decal called RESISTOR-H is a Horizontal Resistor Decal, whereas RESISTOR-V is a Vertical Resistor Decal. You may assume, therefore, that the Decals exist and, therefore, we will first work on ELECTRICAL items. (Alternately, go to Appendix B and search the Listing for the Decal.)

Select F1 PART TYPE and in response to the prompt:

ENTER PART TYPE >

respond R1/4WW. You will be informed that R1/4WW does not exist and asked do you want to create the Decal. Select Yes, then ELECTRICAL and proceed to complete the Electrical Text data.

Part Type
This is already entered for you as R1/4WW.

Prefix
We do not need a Prefix.

Logic Family
Enter RES. The significance of entering RES is that any time this part is used, the Reference Designation will be started with an "R." This is determined by the PKGTYP.DAT file.

PCB Decal and Alternate
In ***PADS-PCB***, the Decal for a 1/4W Resistor is also called R1/4W, therefore, enter R1/4W.

Part Attributes
Part Attributes can be used to display and plot other information, as

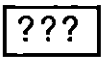
described on page 9-4. For example, a Part Attribute can be used to display and plot the ohmage of the Resistor. Since we will want to do this with a Resistor, enter VALUE: ??? on the Third Attribute Line.

Attribute Lines have a special significance in the instance of Discrete Parts, such as Resistors or Capacitors, wherein you may wish to display at the CRT, plot on the drawing and list on the Parts List, certain values such as OHMS, Farads, etc. Specifically, Attribute Line #3 can be used to assign these values. When so used, in Attribute Line #3, enter the word "Value" followed by three question marks, thus VALUE: ???.

In the Entry "Displayed Attribute Line," enter 3 to indicate that Attribute Line #3 is to be displayed, i.e., VALUES: ???.

The fifth Attribute Line can be used for Tolerance, in a manner similar to Value. If the Tolerance of the Part is desired, enter TOLERANCE ??? on the fifth Attribute Line.

When this particular part is brought up and added to the Schematic on the CRT using ADD PART. You will note on the Gate Decal for the Part, that the Displayed Attribute Line will show:



You may set this Value as desired using the EDIT TEXT routine. For example, if you have two Resistors on the Schematic and you wish one to be a 1K Resistor and the second one to be a 10K Resistor, utilize the EDIT TEXT routine and select the ???. The Part Text Pop-up Window will be displayed. Pick the "VALUE" Line and the VALUE: ??? will be shown in the prompt line. Edit the prompt line to read 1K and the menu will change to:

EDIT TEXT
1 ALL PARTS
2 THIS PART

Inasmuch as you want only this part to be a 1K Resistor, select F2 THIS PART. Repeat for the second Resistor except change to 10K.

You have a similar capability for TOLERANCE on a Part. On Attribute Line 5, enter TOLERANCE: ??? and display Line 5. When used on the Schematic, use EDIT TEXT to set the ??? to the correct Tolerance.

NOTE: The above editing of VALUE and TOLERANCE is NOT done in the Part creation phase, but rather while you are creating the schematic. The above information is provided at this point simply to show how the features would be eventually used.

Displayed Attribute Line
Since we will want to display and plot the Resistor ohmage value, enter a 3 in the first blank. This will display and plot the ohmage value that we enter when the Resistor is added to the Schematic, after having used EDIT TEXT to change the ??? to the correct ohmage. If you wish to display TOLERANCE, enter 5, for Line 5, in the second Displayed Attribute Line, and make sure to double check that they are displayed at the EDIT TEXT level of the schematic.

Non-Numeric Pin Number
Resistors use Numeric Pin Numbers (although suppressed at the CRT and on the plot), therefore, leave the Default as No.

Number of Signal Pins
A common Resistor will NOT have Power and Ground, therefore, leave the Default of 0.

Number of Gates
There will be a single Gate or Decal, therefore, leave as 1

Type
Enter 0 - Not Swappable.

Gate Decals and Alternates
Enter RESISTOR-H for the Primary or Horizontal Decal, followed by a space and RESISTOR-V for the Alternate or Vertical Decal.

When complete, check your Text File and use COMPLETE. After a few seconds, an Error Warning indicating that the Decal Gate Pin Numbers have not been assigned. Your job now is to complete the

Gate Decal.

Select F2 MOD GATE. Then in the Gate Decal Pop-up Window, select RESISTOR-H. First, assign the correct Pin Numbers, using F3 TERMINALS and F4 SET PIN NUMBER. Notice that the Resistor Decal does not have Pin Numbers displayed. The reason for this is that in the DECAL routine, the Electrical pin numbers have been turned OFF for Resistor Decals.

Leave Type as Undefined.

Regarding Swappability, if you do not wish Pins on Resistors to be swapped, leave Swap=0. If, on the other hand, you wish Resistor Pins to be swapped (swapping, will shorten the connection length), change Swap to Swap=1 on both pins.

Select F9 COMPLETE to finish the modification of the Gate.

Now examine the Alternate Decal. Note that the Pin Number and Swappability as set up in the Primary Decal have been duplicated in the Alternate Decal.

When you are satisfied with both the Electrical and Gate information, check the Part using F8 CHECK PART. When the Part is checked and no errors are found, save the Part.

Now bring up the Part on a trial Schematic and use it and satisfy yourself that it has been made correctly. Enter the Resistor three times on the trial schematic. Let's say that you wanted the first entry to be 1,000 ohms, the second entry to be 10,000 ohms and the third entry to be 100K. This is accomplished by using the EDIT TEXT routine and changing the Attribute to read the correct ohmage on each of the three usages. Take care to set your Pin Numbers to OFF to insure that they are not displayed or plotted.

Before finishing your Resistor, check the PKGTYP.DAT file to insure that the category RES has a Reference Designation whose prefix is R.

9.3 CONNECTOR

In this section of the tutorial, we will create a new Connector along with its Decal.

Generally speaking, the creation of Connectors is the same as the creation of Parts, except that in the process one allows for the fact that when a Connector is brought on to the sheet of a Schematic, it is usually brought up at the Pin Level, rather than the entire Component Level and, secondly, the Connector Part is generally not shown on the sheet of the schematic, as would be a 74LS08 or other normal part. To do this, we have a routine to create Connectors that is slightly different from Parts.

In this exercise we will create a Connector, including its Schematic Decal, but not the PCB Decal. This connector will be a 10-pin Edge Finger Connector.

Regarding the Naming convention for Connectors, you should generally name a Connector in such a fashion as to use the Name to identify the type or characteristics of the Connector. In this particular case, we will call the connector CON/10P/ED. This nomenclature is for Connector, 10 Pins, Edge Finger. We will create two Pin Decals: one for Input Pins and one for Output Pins. The Input Decal will be called CON/I and the Output Connector will be called CON/O.

Inasmuch as these Decals do not currently exist, let us first create Decal CON/I and CON/O.

Be in HIER/LIB and select F2 CONNECTOR. CONNECTOR is used to create and modify Connector Parts. Connectors are normally shown at the Pin level on the schematic rather than at the Part level, hence a different subroutine is used to create Connectors.

NOTE: F5 GATE DECAL will also work in the creation of Connector Decals--but due to the fact that Gates contain Pin Decals (unlike Connectors), you will receive warnings about No Pin Terminals in your newly made Connector Decal at the completion of the Decal. We, therefore, suggest you use the F2 CONNECTOR subroutine.

In response to the prompt:

PART DECAL NAME (CR = TEST)

respond CON/I. You will be told that CON/I does not exist, and asked if you wish to create it. Respond Y, select MOD GATE, and the Reserved Text Block for Reference Designation appears at the CRT.

The following menu appears:

OTHER MENU
HIER/LIB
1 2-D LINES
2 ADD TEXT
3
4 SETUP
5 COPY
6 MOVE
7 DELETE
8 ORIGIN
9 COMPLETE
0 EXIT

Use F1 2-D LINES to create the Input Symbol as follows:

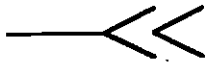
To give you control, change the Grid to a Grid of 50 by typing in G50. You can get finer control with a smaller grid, but for this exercise, it's not necessary. You will probably want to use the dot grid on the screen to control the size. As a suggestion, make the Width of your Input arrows 100 mils. As a further suggestion, after you have made your first arrow, rather than creating the second arrow, use F5 COPY and copy the first arrow.

When you have your Decal created, place the ORIGIN at the point on the Decal wherein you want the connection to connect to the Connector Decal. This is done by placing the cursor at that location and selecting F8 ORIGIN.

Next, using F6 MOVE, position the Reference Designation box (U999A) at whatever location relative to the Symbol you wish the Reference Designation and Pin Number to appear on the Schematic. As a suggestion, although you could show the Connector Package Type and one or two Attributes, it is highly doubtful that you will want to do so, therefore, use NAME DISPLAY (in F4 SETUP, F4 NAME DISP) and turn off the Package Name, Part Attribute #1, Part Attribute #2, and leave Reference Designator ON.

At this time, use F9 COMPLETE and save the Decal using the name CON/1.

Repeat the sequence for the Output Connector Symbol (CON/0), except changing the Symbol to as follows:



This completes the creation of the two Decals that we will use with our Connector. Let us proceed now to creating the Connector Part.

Select F2 CONNECTOR and in response to the prompt for the Name, enter CON\10P\ED, and indicate you want to create the new part. Next, select F1 ELECTRICAL.

You will notice the Connector Name has been entered for you.

Under Number of Connector Pins, enter 10.

NOTE: This is the number of pins on the actual PCB Connector--Not the Decal.

Logic Family
Notice that the name of the Logic Family has been entered for you as CON. This will insure that the Reference Designations will always start with J (see discussion on PKGTYP.DAT File).

PCB Decal
If you have ***PADS-PCB***, you would want to name the PCB Decal the same as the Part Type (CON\10P\ED); alternatively, if you are not using the ***PADS-PCB*** Board CAD System, enter the PCB Decal that you would normally use for 10-pin connectors.

Number of Part Attributes
Leave this 0 (unless you wish to display/plot other than the REF DES and Pin No).

Displayed Attribute Lines
Leave this 0 (unless you wish to display/plot other than the REF DES and Pin No).

Non-Numeric Pin Numbers
Leave this at the Default N.

Number of Connector Pin Types
Enter 2 since there will be a Primary (CON/I) and an Alternate (CON/O), and under the Pin Type Connector Pin Decal Entry, enter U for Pin Type and for the first Connector Pin Decal, enter the Primary Decals which, as you will recall, is CON/I. For the Alternate, enter U for PTYPE and CON/O for the Decal on the second Pin Type data line.

NOTE: With Connectors, the Alternate Gate is on a separate line from the Primary, rather than on the same line as with "PARTS."

Use F9 to COMPLETE the Electrical section. This organizes the data for storage.

Next, select F2 MOD GATE.

In response to the prompt:

ENTER GATE NUMBER TO EDIT (CR = 1)

enter 1 <CR>. Then select F9 COMPLETE. This will add the pin information to the Decal. Repeat this process for Gate 2.

Upon completion, run a check of the complete Connector Part, including the Electrical and Gate information, using F8 CHECK PART. If there are no errors, save the Part to the Part Library, using F9 SAVE PART.

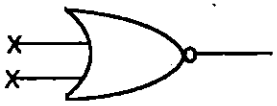
As a final "proof positive test," call up the Part on a sheet of the Schematic and insure that it performs and is presented as you would

expect.

9.4 CREATING A NEW GATE DECAL

In this section of the tutorial, we will create a New Gate Decal using the GATE DECAL routine. (You were briefly exposed to this subroutine above when you made the Decal for the Connector.)

The Decal that we will make will be a Two-Input Positive Gate as shown below:



and its DeMorgan Equivalent shown below:

NOTE: A word on the System Grid used for IC Decals: The IC Decals included in the library have, for the most part, been created with a Grid of either 50 or 100 for the Outline, and 100 for the Terminals. This is of great value during the creation of a schematic where the Grid should be set to either 100 or 50, since it will result in the connections "snapping" to the terminal.

Select F5 GATE DECAL and in response to the prompt:

PART DECAL NAME (CR = TEST)>

enter NOR22 (The real 2-Input NOR Decal is in the Library as NOR2, therefore, our use of NOR22 will NOT endanger the real Decal.)

DECAL NOR22 DOES NOT EXIST -
DO YOU WISH TO CREATE DECAL (Y/N)?

Respond "Y" to the creation of New Decal.

CREATE BOX SYMBOL AUTOMATICALLY (Y/N?)

is presented. Inasmuch as in this exercise we wish to create this Symbol, respond N.

The four Reserved Boxes for Text are shown at the CRT.

The menu structure that allows you to create the Part is presented. Using 2-D LINE, draw a figure approximately as shown below:

It is suggested that you use a grid of 2. This can be obtained by using the Modeless Macro Command of typing G2.

In the process of creating the figure, you will want to be aware of the size or scale of the figure. Normally in the ***PADS-LOGIC*** Library, Gates are created 4/10" high. It is suggested that you use the dot grid shown at the CRT set up to 400 as a guide for the size of the Gate.

After you have drawn the figure shown above, use 2-D, MODIFY, MOVE ARC to reconstruct the polygon to represent a NOR Gate. Use 2-D LINES, NEW CIRCLE to create the small Negation Bubble. Once you have completed the figure, tweaking is best done by MODIFY and dragging the 2-D Line to the new location, then using COMPLETE to finish.

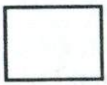
Once you are finished with the Outline, you will want to add the Terminals, using F3 TERMINALS. It is suggested that when you do this, you utilize a grid of 100.

In general, the IC Parts shown at the Gate Level within the ***PADS-PCB*** Library, have the Reference Designation located within the Gate Outline. When shown at the Part Level, the REF DES is shown above the Part Outline. Using MOVE, move the Reference Designation Reserved Text Box to within the Outline. Move the Package Type to immediately below the Outline. When satisfied with the Decal, use F9 COMPLETE. Observe carefully to see that it has been saved, and to double check, use GATE DECAL to call it

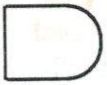
back and examine it.

Next, use the same procedure to create the DeMorgan Equivalent as shown below:

Use the name NORDMM (to differ from the "real" Decal of NOR DM). Use 2-D LINES to create a figure as follows:



Then use MODIFY, MOVE ARC to obtain:



Add the Terminals, complete and save.

PRACTICE: Follow the procedure outlined in Paragraph 9.1 to create a part using the Gate Decal you just created. Call up the Part and create a small Test Schematic, as a final check.

9.5 THE PKGTYP.DAT FILE

Reference Designations (REF DES) for a part are automatically assigned using as the source, a file called PKGTYP.DAT. This file is included with the software, however, it can be edited by the user.

Basically, the file contains the stem of the REF DES of a part that, in turn, is associated with the Logic Family entered in the Electrical section of a part. While creating the Electrical data for the part, i.e., RES will use an R as the stem of the REF DES. TTL, CMOS and ECL logic families will use a U as the stem.

The contents of this file, as shipped with the software, is as follows:

```
RES  R
TTL  U
CMO  U
ECL  U
ANA  U
CAP  C
DIO  D
ZEN  Z
TRX  Q
XFR  T
CHO  L
CON  J
RLY  RL
SCR  TH
TRC  TH
SWI  SW
```

ADDED
IND L
POT VR
RNT RP
XTL Y
JMP JP
BAT B
TPT TP

NOTE: An undefined part will, when brought into the schematic, request YOU to provide the stem for the REF DES.

You may print the file using the standard DOS PRINT Command.

You may edit the file using the PEDIT Editor.

9.6 DELETING A PART

From time to time, you may wish to Delete a part or a Decal from the Library. Deletion is performed with the PLIB Utility which, in turn, is accessed from the PADS Shell.

Let us delete the \022 IC part you just created, as an example of using PLIB.

First, go to Part III, the Reference Manual, Section 13.2 and read the section on PLIB.

1. Go to the PADS Shell and select the PLIB port.

2. Select LIBRARY: Since you created the \022 part, it will be in

the Users Sub-Library. Enter U.

3. Select DATABASE: Since we wish to delete a Part, select P for Part Type.

4. Select ACTION: Since we want to delete an item, select D.

5. Select ITEM NAME: Enter \022 <CR>. The ASCII FIle for /022 is displayed. Respond to:

ARE YOU SURE (Y/N?)

with "Y" and the \022 part is deleted from the Library Decal (i.e., the ELECTRICAL and GATE information, but not the Original General Decals.

6. To leave PLIB, use Bye (B) repeatedly, then at the Top level, respond to:

ARE YOU SURE (Y/N?)

with "Y" and you will exit PLIB and return to the PADS Shell.

TUTORIAL TEN:

FILES MANAGEMENT

10.1 OBJECTIVE

This tutorial discusses the control and storage of the files created with the system. It is absolutely essential that you understand and follow the practices and procedures described below. Failure to do so will result in lost data, in which you have invested a considerable amount of your time.

10.2 JOB DIRECTORY LISTINGS

As you have been storing design jobs either in Binary or in ASCII, they have been written on the hard disk in a subdirectory of PADS called LOGFIL. From time to time, you will want to know what design files are stored, whether or not a specific design is on your hard disk, how much additional remaining space you have on your hard disk, etc. To do this, you will need to obtain a CRT listing of the subdirectory LOGFIL.

To obtain a listing of all files in the LOGFIL subdirectory, proceed as follows:

To the display: C:\>
Enter: CD\PADS\LOGFIL
The Response: C:\PADS\LOGFIL indicates that you are now in the LOGFIL subdirectory containing all Job files.
To the Response: C:\PADS\LOGFIL>
Enter: DIR <CR> for the Directory Listing.

A listing of the contents of the LOGFIL subdirectory will appear at the CRT, along with the date of the file, size of each file, number of files and free space remaining on the hard disk. It is a good practice to examine your LOGFIL subdirectory contents on a daily basis and delete those files that have outlived their usefulness. Before deleting a file, however, be sure you will never need the file!

PRACTICE: Make a Directory Listing of your hard disk. It should

contain, as a minimum, DEMO.SCH, SELFTRAN.SCH, CAE.SCH, as well as files you have made during the preceding tutorials.

10.3 JOB CONTROL PROCEDURE

You have recognized that in addition to containing your **PADS-LOGIC** Program, your hard disk will also contain an ever growing number of both Binary job files and ASCII Schematic files. In this section, we will discuss the Control and Management of these files so that the loss of valuable data is minimized and you know where your designs are at all times.

During the process of creating a schematic, we would suggest that in addition to the Automatic Backup made every few minutes by the system, you should make a Binary Job Save approximately every hour, or at the beginning and completion of each significant event, i.e., immediately after the completion of Part Symbol placement, every hour or so during adding connections, and a final file, upon completion of the Schematic and ready to Post Process. This final file becomes the Archival file.

This may sound like a lot of files to make but, remember, disk space is cheap as compared to your time; in the process of capturing the Logic of your design, you can probably make in the order of 25 or more Schematic Job Saves on your hard disk, and all but one of these files are considered temporary and will eventually be deleted before you start your next schematic. (You will save the final file as the Archival file.)

To summarize, the general philosophy is to be liberal with your Saving of Binary Job files. In this fashion, if something should happen to your work in process, i.e., it becomes contaminated or you have designed the schematic into a "blind alley," then you do not have to go back to the beginning and start all over again. You simply go backwards in the stored job sequence until you find the appropriate job file and start again from that point. In this regard, it is important to keep a written Job Save Log during the design of a board in order to assist you in identifying the contents of each Job Save file.

After you have made several Job Saves on a design, it is difficult to remember the contents of each Save. Therefore, as your design progresses, we suggest that you keep a written Job Save Log similar to that shown in the figure below as follows:

JOB SAVE LOG				
Job Name	Date	Time	Designer Initials	Job Description

SUGGESTED JOB SAVE LOG

10.4 DELETION OF FILES

After the schematic is post processed, we strongly recommend that you remove all of the interim files created during the process of design that are on the hard disk, leaving only the last or completed file for use as the Archival file. This is accomplished as follows:

First, just in the event you make an error during the deletion of interim files, PLAY IT SAFE by first copying ALL files associated with the schematic on the hard disk in the LOGFIL directory to a floppy disk. Keep this floppy until the deletion of all unnecessary files on the schematic is complete, for use in restoring the hard disk, if necessary.

To delete files from the LOGFIL subdirectory, use the DOS command of DEL or Delete. Be in the PADS\LOGFIL path and enter the Delete command along with the Job Name and Extension:

C:\PADS\LOGFIL>DEL XYZ.SCH <CR>

HINT: Before deleting any files, make a hard copy Directory Listing of LOGFIL. Check off only those you want to delete. Double check your decision! After you have deleted the interim files, make a listing of the directory to insure that you have deleted what you intended to delete and have left on the disk, those files that you wish to keep.

PRACTICE: Delete a file that you do not need to save.

10.5 DAILY BACKUP

We recommend that at the end of your shift, you always save all the Job files created that day on the job. This is an "insurance" policy against a system failure or human oversight that deletes, overwrites or otherwise destroys your effort that exists on hard disk. Should such an unhappy event occur, you have your floppy disk of your work, which you load back onto hard disk and proceed.

10.5.1 MAKING A BACKUP FLOPPY

To make a Backup, access the LOGFIL subdirectory by entering:

CD\PADS\LOGFIL

The response should be:

C:\PADS\LOGFIL

to indicate that you are in PADS\LOGFIL subdirectory. Use the following command in response to C:\PADS\LOGFIL.

Enter: **COPY XXX.SCH A: <CR>**

The correct translation of the above is: You are currently in the subdirectory PADS\LOGFIL on the hard disk. You wish to copy

from this subdirectory a file called XXX.SCH, and copy it to the floppy disk contained in Drive A and you wish it to continue to use the name XXX.SCH. The disk light will light up while the transfer is occurring and the message:

1 FILE COPIED

will occur. You should at this time obtain a directory of the floppy disk in drive A by entering the command DIR A: <CR> to insure that the file XXX.SCH has been placed on the floppy.

The DOS Wild Card (*) feature can be of assistance in reducing the entry of data.

Examples:

1. You wish to copy ALL files in \PADS\LOGFIL to floppy

In response to: A:\>
Enter: COPY *.* A: <CR>

2. You wish to copy all jobs with the extension .SCH to the floppy

In response to: A:\>
Enter: COPY *.SCH A: <CR>

3. You wish to copy all the files associated with Schematic Name XXX to the floppy

In response to: A:\>
Enter: COPY XXX.* A: <CR>

Take care to follow the above format including spacings. Note that DOS always copies FROM a device TO another device, that is, the source device is always listed first in the copy command. As soon as the files are copied, obtain a listing of what is on the floppy disk by entering DIR A: <CR>.

In the event there is not sufficient room on the floppy to copy all the files, use a second disk and copy the files not copied on the first disk.

PRACTICE: Copy all the files you have made for SELFTRAN onto floppy.

10.5.2 FILES ON FLOPPY DISK

So you have copied all your files created during the day onto a backup floppy disk. Good! Now, make a printout (Directory) of the contents of the floppy, and label the disk, and protect this disk like the "gold" it is!

PRACTICE: Make a printout of the files on the floppy disk created above.

10.5.3 USING THE FILES DISK

Let's say you come to work the next day and you find that half of the files you had created on hard disk the day before, have been erased by the night shift designer. How do you get yesterday's files on floppy disk back on the hard disk?

Enter the PADS\LOGFIL subdirectory such that the display is:

C:\PADS\LOGFIL>

and have the backup disk in drive A.

You enter: **Copy A:*.* <CR>**

This will copy ALL the files on the floppy over to the hard disk and place them in \PADS\LOGFIL. Any existing files on the hard disk that are also on the floppy disk will be overwritten and any missing files on the hard disk will be copied from the floppy disk, such that at the end of the transfer, all files on the floppy disk will now be on the hard disk. Use DIR to check.

Alternatively, you could have copied only the erased files.

10.6 ARCHIVAL FILES

The file on the hard disk representing the finished schematic and the file used for plotting the finished schematic should be considered the

Master or Archival file and placed in your company's archives. It should be controlled with exactly the same care your company's Configuration Management and Documentation Control system controls a Master drawing. To do this, we suggest you use floppy disk media if you do not have a Mag tape on your system or access to a Mag tape. Use only High Quality floppy disks.

Copy the final file used for post processing onto two different floppy disks: Archival Disk 1 and Archival Disk 2. Label them accordingly. We suggest you do not put jobs from other schematics on these disks. Dedicate a disk to the files associated with one schematic, including all its revisions. Disks are inexpensive and the "one schematic--one disk" procedure will greatly simplify your administration. Store the two disks safely in two separate locations.

The two separate locations should be in two different buildings or sufficiently separated so that if a catastrophe such as a fire destroys one set of Archives, it does not destroy the other.

At some time in the future when a revision is made to the particular schematic, copy the database from the archival floppy, read it into the hard disk and make the revision, taking care to uprev the file name, i.e., XXXA.SCH, XXXB.SCH, etc. Store the revised file on the same two archival floppies where the original job was stored.

In this fashion, you will be able to dedicate a floppy disk to a single schematic and all its revisions, thus greatly simplifying your record keeping and retrieval of work you have performed, yet providing protection to your files.

The DOS copy commands for copying from hard disk to floppy disk are the same as for backing up your work files.

When you have successfully copied your Archival file from hard disk to a floppy (obtain a listing of the floppy disk, insuring the file has been copied), read the file back onto the hard disk (using a different name) and check that it can be called up in **PADS-LOGIC**. You can then delete the Archival file from your hard disk, thus freeing up more hard disk space.

PRACTICE: Make an Archival File of the final SELFTRAN

schematic. Copy it to a floppy. Bring it back onto the hard disk and issue it as useable.

A NOTE OF CAUTION: Occassionally, in the process of transferring from hard disk to floppy, a data error may occur, thus invalidating the copied disk. To guard against this, each time you transfer to the floppy, immediately transfer it back to the hard disk and attempt to call it up using the JOB IN subroutine. In this fashion, you can be assured that the database for your completed schematic has been successfully transferred to the floppy and can be retrieved at some time in the future.

In the event you have access to a Mag tape on your system, you may want to use this media as your Archival medium rather than floppy disks. In this case, follow the same procedures of making two different Archival tapes, reading them back into the system and bringing them up at the CRT to insure that the copy has been made successfully and store the two mag tapes in separate locations.

10.7 STORAGE SPACE

You should be aware of the storage space required for the database of a schematic, together with the capacity of a floppy disk. The ***PADS-LOGIC*** database for a schematic is very efficient, approximately 1kb of disk storage is required for each equivalent 14 pin DIP on the schematic. Example: a schematic with 100 equivalent 14 pin ICs would require approximately 100kb of storage. Thus, if you are using a CPU with 1.2Mb floppy disks and designing a 100 equivalent 14 pin IC schematic, you can store about 11 files on a floppy before the message DISK FULL is given.

What do you do if you are transferring a series of files to a floppy disk and you receive the DISK FULL message? Don't panic. You should always have extra formatted floppy disks available. Make a directory of the full floppy to determine what is on the disk. The last file will be the file that generated the DISK FULL message. It probably is no good since it is incomplete. Delete it from the floppy. Put the new formatted disk in the drive and copy that file and any other files not copied, onto the second disk. Make a directory listing of both disks to be sure you have copied all the files you intended to save.

PART III

THE REFERENCE MANUAL

THE CRT PRESENTATION	CHAPTER ONE
THE IN/OUT ROUTINE	CHAPTER TWO
THE ADD PART ROUTINE	CHAPTER THREE
THE ADD CONNECTION ROUTINE	CHAPTER FOUR
THE EDIT TEXT ROUTINE	CHAPTER FIVE
THE COPY ROUTINE	CHAPTER SIX
THE MOVE AND DELETE ROUTINE	CHAPTER SEVEN
THE SETUP ROUTINE	CHAPTER EIGHT
THE ADD BUS ROUTINE	CHAPTER NINE
ADD TEXT ROUTINE	CHAPTER TEN
2-D LINES ROUTINE	CHAPTER ELEVEN
GROUP ROUTINE	CHAPTERN TWELVE
THE PARTS LIBRARY MANAGER (PLIB)	CHAPTER THIRTEEN
THE PARTS LIBRARY	CHAPTER FOURTEEN
THE HIERARCHICAL ROUTINE	CHAPTER FIFTEEN
THE MACRO ROUTINE	CHAPTER SIXTEEN
LIBRARY PARTS	APPENDIX A
DECAL PLOTS	APPENDIX B
NETWORKING	APPENDIX C

PART III

PADS™-LOGIC

REFERENCE MANUAL

Chapter 1

THE CRT PRESENTATION

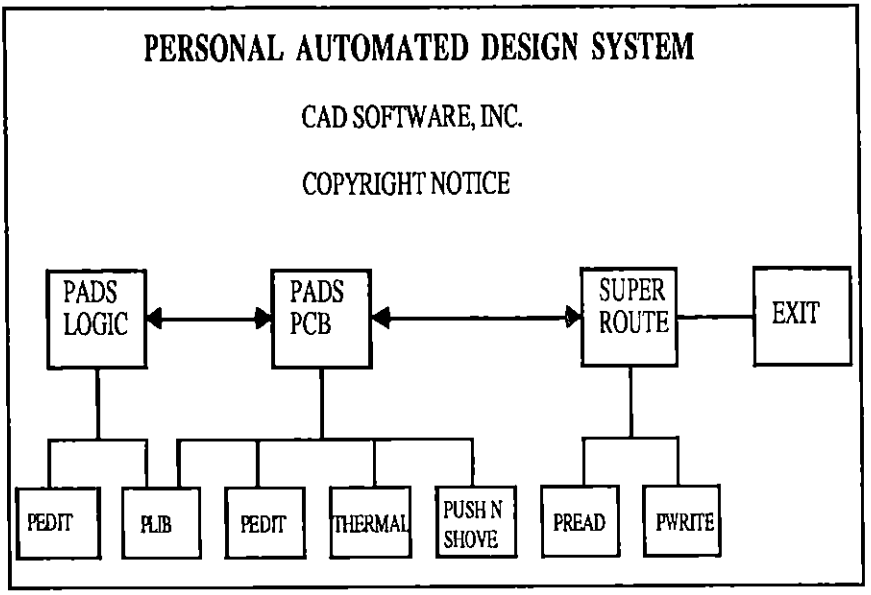
1.1 PURPOSE

The purpose of this chapter is to familiarize you with the CRT presentation, its graphics and text contents, and how to manipulate them, when in ***PADS-LOGIC***.

1.2 *PADS-LOGIC* PROGRAM INITIALIZATION

After the ***PADS-LOGIC*** program has been loaded onto the hard disk, the LOGIC program is initialized as follows:

Display:	C:\>_
YOU TYPE:	**PADS<CR>** NOTE: (<CR> is Enter)
Response:	The response will be the PADS SHELL by which the various PADS programs are called.. The SHELL has the following options plus EXIT as shown below:



PADS-PEDIT is a Special Text Editor that supports ***PADS-LOGIC***.

PLIB is the ***PADS-LOGIC*** Parts Library Manager.

PADS-PCB is the CAD Software, Inc. Printed Circuit Board CAD System. It is described in Volumes I and II, ***PADS-PCB*** Program Reference and User Manuals.

PADS-THERMAL is an option to ***PADS-PCB*** that provides an analysis of the Thermal results of different component placements on the Printed Circuit Board.

PADS-PUSH N'SHOVE is an Interactive Autorouter option to ***PADS-PCB***. It is described in an Appendix of Volume II, **PADS-PCB** Reference Manual.

SUPERROUTE is the Rip-Up and Reroute Autorouter option to ***PADS-PCB***. It is described in Appendix A-4, Volume II, ***PADS-PCB*** Reference Manual.

To initialize the ***PADS-LOGIC*** Program, proceed as follows:

YOU — Select the **PADS-LOGIC** box with the mouse cursor.

RESPONSE: — The copyright screen will appear together with the software version number.

Additionally, the amount of DOS memory available for Logic Capture is presented. Press any key and the CRT presentation of Figure 1.1 will appear.

1.3 *PADS-LOGIC* CRT PRESENTATION

When the ***PADS-LOGIC*** Logic Capture Program is loaded into memory, the CRT presentation as shown below will appear after approximately 3 seconds following the copyright notice and the pressing of any key.

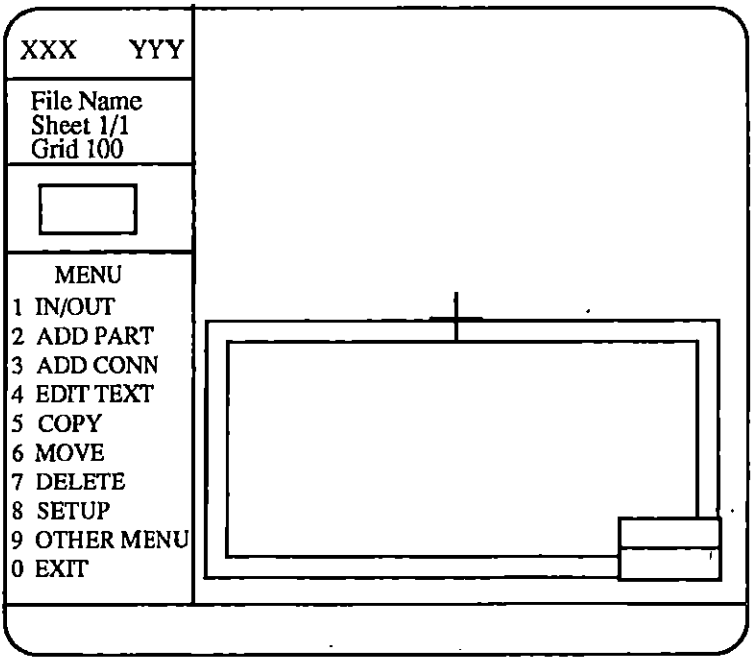


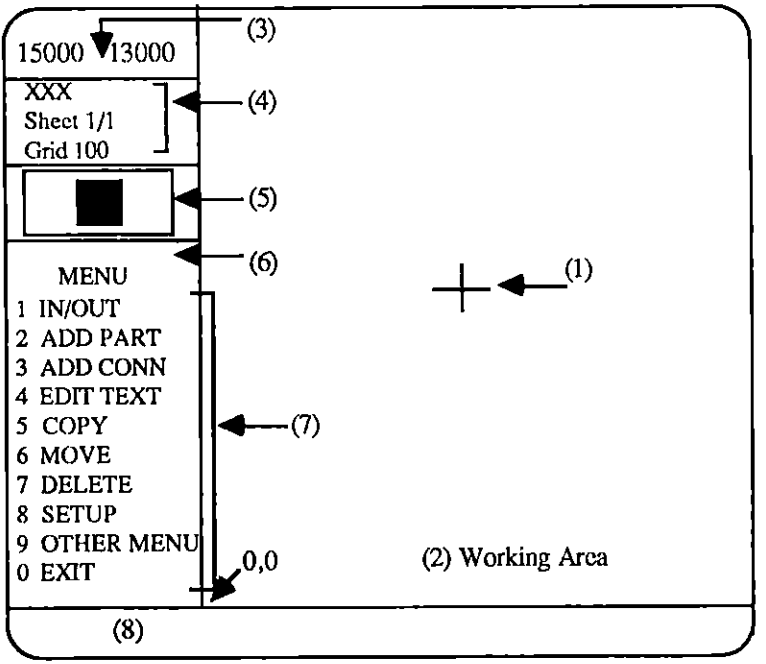
Figure 1.1 - INITIAL CRT PRESENTATION

In this manual, <CR> stands for the Carriage Return or Enter key. Each time you use the keyboard for entering a job name, a command, or any alphanumeric text string, etc., you enter <CR>. Striking the Carriage Return key indicates to the computer that your command or input stream is completed and can be acted upon.

HINT: If, in the use of your ***PADS-LOGIC*** Program, you have entered the command string and nothing happened, you may have forgotten to hit the Carriage Return key.

1.4 CRT FEATURES

The figure below and the following discussion, identifies and describes the different items presented on the CRT Terminal.



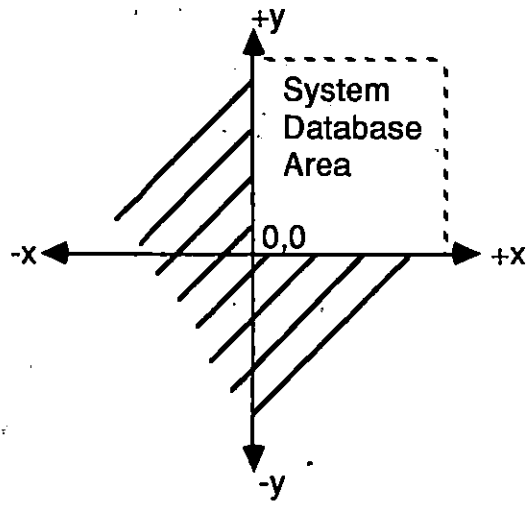
CRT FEATURES

1.4.1 CURSOR (Item 1)

The cursor consists of a white cross that highlights so as to stand out in the midst of both angled lines and orthogonal lines. The cursor movement is generated by moving the mouse.

1.4.2 WORKING AREA (Item 2)

The working area is the portion of the CRT where the Schematic or Logic Capture is created; it is your Drafting table. The working area, or database, uses the upper right quadrant of a Cartesian coordinate system as shown below:



DATABASE AREA

The System Origin (0,0) is at the lower left corner of the database coordinate system, which corresponds to the lower left corner of the CRT. When a drawing sheet outline is present at the CRT, the system origin is at the lower left corner of the drawing sheet. All cursor positions within a drawing sheet have positive values.

The system database and Working Area have a maximum X and Y value of 64000 in 2 mil units. The database is a square area 64.0" x 64.0" on a side and all linear dimensions used in the system are in mil units.

The database contains a snap grid to which parts, conductors and text are "snapped" when placed. The Default value for the snap grid is G=100 (0.100"). This may be changed to any value between G=2 and G=2000, in mil increments.

To aid part construction, part placement, conductor spacing, and text placement, a variable dot grid is provided. This visible grid provides a reference to size and distance when working at higher magnification levels.

In order that the CRT Display not be covered with "dots," the visibility of the dot grid is a function of the dot grid spacing as determined in the Routine SETUP, and the Zoom and Expansion level of the CRT.

Within the Working Area is a Drawing Sheet. (D-Size is Default.) Should you not want to use a D-Size Sheet, you may change to a different size, using SETUP, PARAMETERS.

1.4.3 CURSOR POSITION INDICATOR WINDOW

(Referenced to System 0,0)
(Item 3) (White)

The two numbers located in the upper left hand corner of the screen show the X and Y position of the cursor in mils with respect to the System 0,0. When a Drawing Sheet outline is in memory and displayed at the CRT, System 0,0 is at the lower left corner of the Drawing Sheet. Thus, the cursor position when within the drawing sheet will always be positive. When the database grid is at G=50, the cursor position readout is in 0.050" increments, and the positional location of parts, conductors, text and lines is to the nearest .050"; when G=100, the readout is in 0.100" increments, etc.

1.4.4 DRAWING NAME, SHEET AND GRID DATA

(Item 4) (Purple)

The drawing name is shown at the top of the purple window, when a drawing is in memory.

Sheet X/Y indicates Sheet X of a total of Y sheets.

Grid XX indicates the database is set to a grid of XX thousands of an inch, i.e., the System Grid. Cursor minimum movements are equal to the grid value . The grid can be changed by typing G XX <CR> where XX is the new Grid value (Minimum: 2; Maximum: 2000). (Within these limits, the Grid may be set in 1 Mil increments.)

1.4.5 ACTIVITY WINDOW
(Item 5) (Yellow)

This multi-line window is "activated" for certain activities. Either the "Postage Stamp" feature or Text Data is displayed.

A "Working" line in the CRT Activity Window becomes active when the CPU is dedicated to a task such as making a backup job file, entering a drawing file, etc. When the CPU is thus dedicated, the word "WORKING..." is displayed.

When lines or text data are not displayed in the Yellow Window (the normal condition), the "Postage Stamp" feature is displayed. The Postage Stamp feature displays either the schematic sheet outline or the 64" x 64" database limits together with the "Postage Stamp", a blue color filled rectangle representing that area displayed at the CRT, relative to either the database or a sheet of the Drawing being displayed.

When a drawing is present in memory, the sheet outline is presented in its true shape and size relative to the postage stamp area. When a sheet outline it not present, the blue square represents the 64" x 64" database limits.

An excellent way to keep your orientation as to where you are on a sheet as you move the CRT display contents around, is to observe the postage area in relationship to the sheet outline, while repeatedly pressing the Right, Left, Up, Down numeric keys, or Auto Panning.

1.4.6 COMMAND MENU TITLE BLOCK
(Item 6) (Blue)

This multi-line Title window is used in conjunction with the Command Window (Item 6) and identifies the selected major command routine title, its first subroutine, and then the next lower level or option. For example, if in the major routine of IN/OUT, in the ECO subroutine, the title window would display:

IN/OUT
ECO

The window changes with changes of the Command Menu.

1.4.7 COMMAND MENU WINDOW
(Item 7) (Blue)

The Menu Window displays either the major routines, their subroutines or options as used to control the system during the design activities.

The ***PADS-LOGIC*** Program uses a Hierarchical Command menu structure. The highest rank in the hierarchy is the Major Routines, of which there are nine different Routines, plus EXIT, as follows:

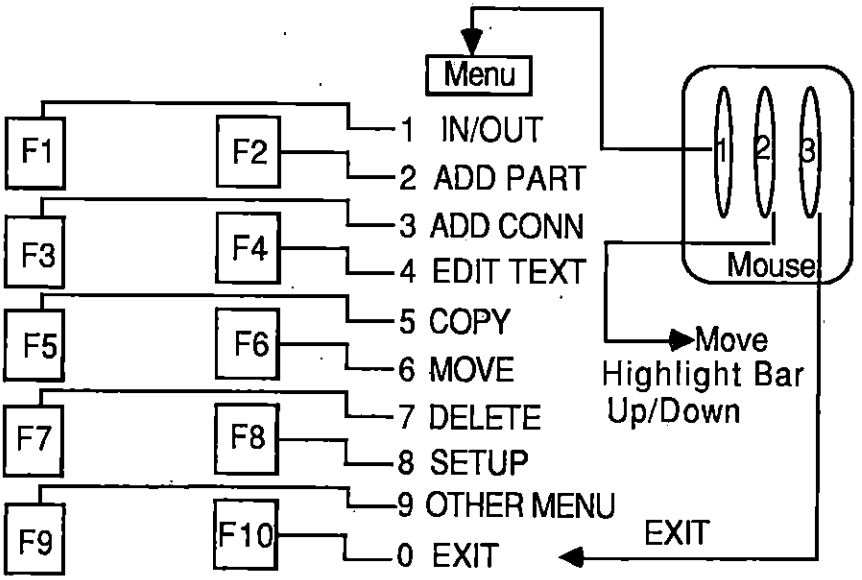
1 IN/OUT
2 ADD PART
3 ADD CONN
4 EDIT TEXT
5 COPY
6 MOVE
7 DELETE
8 SETUP
9 OTHER MENU
0 EXIT

When any one of the nine Major or Top Routines is selected, the Major Routines are replaced with the next highest rank Subroutines (the Subroutines associated with the Major routine that was selected).

The Subroutines, in turn, have from one to nine Subroutine Options associated with each of them. When a Subroutine is selected, the subroutine menu is replaced with the Options associated with the selected subroutine.

F10 EXIT, in both the subroutines and options, is used to exit that rank or level and go to the next higher menu rank. For example, if in an option, EXIT will move the system from the option up to the subroutine containing the option. If in a Subroutine, EXIT will move the system from the Subroutine to the Major Routine containing the Subroutine.

F10 EXIT, when at the highest or Major Routine level, permits the "leaving" of the ***PADS-LOGIC*** Program and returns the system to the PADS SHELL.



Function Keys

MENU COMMAND SELECTION

The Command Menu items are selected with either the "F" Function Keys located on the keyboard or the Mouse buttons as shown in the

above figure. Menu routines, subroutines and options, 1 through 10, are selected with the F1 through F10 buttons located on the keyboard.

Alternatively, a three-button mouse (PC Mouse provided by Mouse Systems Corporation, 2600 San Tomas Expressway, Santa Clara, CA 95051, Manager Mouse provided by Numonics, 100 Commerce Drive, Montgomeryville, PA 19836) is especially usable as an input device, and is mapped such that in addition to moving the cursor, its buttons can select any of the ten Command Menu choices. The left hand mouse button (button #1) is mapped to select or activate the routine, subroutine or option over which the Menu Highlight Bar has been placed. The middle or mouse button #2, when continually depressed (while rolling mouse up and down), is used to move the Menu Select Highlight Bar up or down over the menu items to be "selected" with mouse button #1. Finally, the right hand mouse button (button #3) is mapped to the EXIT routine in the same fashion as Function Key F10. If you are using a 2-button mouse, the middle button function is not available and the right button is mapped to the EXIT routine in the same fashion as Function Key F10.

To move the Highlight Bar from menu command to menu command, depress Mouse Button #2 and move the mouse in a vertical motion. Move the mouse until the Highlight Bar is over the desired menu command, then select mouse button #1 to select that menu command. Alternately, vertical movement of the mouse, while the cursor is located within the menu area, can be used to move the highlight bar to different Command Menu positions. To do this, move the cursor until it is located in the Command Menu area after which up/down movement of the mouse, without depressing the middle button, will cause the highlight bar to be moved.

"Clicks" of Mouse Button #2. Using this method, a "Click" of Button #2 will advance the Highlight Bar from Top to Bottom of the Menu Commands, with each click advancing the Highlight one Menu item. To use this technique, you must change your Software slightly. Proceed as follows:

1) Exit from ***PADS-LOGIC*** and return to the DOS prompt.

2) Be in the \PADS directory. If necessary, enter: CD\PADS.

3) Delete the file called "CURSOR." (enter: DEL CURSOR).

When you re-enter ***PADS-LOGIC***, each "click" of the middle mouse button will "step" the highlight bar down to the next menu item.

If after trying this method, you find you wish to return to the original operations, proceed as follows:

1) Exit from ***PADS-LOGIC*** and return to the DOS prompt.

2) Be in the \PADS directory. If necessary, enter: CD\PADS

3) Create a file called "CURSOR," as follows:

Enter: PEDIT \PADS\CURSOR. (This will bring up PEDIT and create a file called CURSOR. (The file will not contain data.)

Select F10 SAVE and then "ENTER". (This will save a file (without data) called CURSOR).

Select F9 QUIT. (This will exit PEDIT and bring you back to DOS.)

When you re-enter ***PADS-LOGIC***, the mouse will function in the original mode.

In summary, Function buttons F1 through F10 can be used, depending upon the choice, to select the ten Command Menu choices. The Mouse, in addition to controlling the cursor, can also be used to select Menu Commands.

The designer will find that a position with the left hand over the Function buttons and the right hand on the Mouse, is the most comfortable operational position, and with practice, the simultaneous selection of the proper routines and the control of the cursor, will become rapid and efficient. Alternately, the designer can use only one hand and the mouse to position the cursor as well

as to select the Menu item. It is suggested that you attempt both methods and utilize the technique best suited to your personal reaction.

1.4.8 MENU MACROS

Five Menu Macros have been provided to permit rapid/simple command movement from one menu level to another menu level, several levels different from the current level. Normally one would repeat the EXIT command to move from a lower level to a higher level, however, if there are several intervening levels, this can be time consuming, hence, the Menu Macros.

1. Top of Schematic Menu Level
To move from any level in ***PADS-LOGIC***/Library, to the top menu level of ***PADS-LOGIC***, simultaneously strike the ALT Key and the T Key.

2. Top of the Library Menu Level
To move from any level to the top of the Library Menu, simultaneously strike the ALT Key and the L Key.

3. Exit to DOS
To Exit the ***PADS-LOGIC*** program to DOS from any menu level, simultaneously strike the ALT Key and the X Key.

4. ECO Menu
To move from any level to the top of the ECO Routines, simultaneously strike the ALT Key and the E Key.

5. Macro Menu
To move from any Level to the Macro Routine, simultaneously strike the ALT Key and the M Key.

NOTE: The Menu Macros listed above are contained in a file in \PADS\LOGFIL called CAE.MAC.

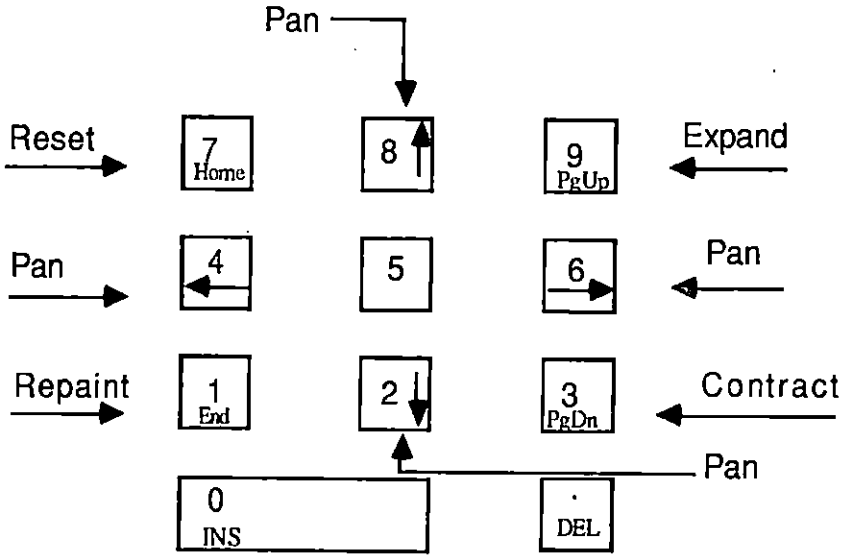
1.4.9 SYSTEM PROMPT WINDOW
(Item 8) (Red)

This single text line window is the primary means of communications between the CPU and the designer. Potential Error Messages are also displayed in this window.

1.5 CRT MANIPULATION

CRT manipulation is the means by which the designer moves the drawing representation around on the CRT, locates it in different areas on the CRT, expands the drawing, contracts the drawing and, in general, manipulates the drawing for the activity in process.

Located on the keyboard is a Numeric keypad containing Numeric keys 0 through 9 and the decimal point ".". All of these keys, except for the "5" key, are used for manipulating the CRT as shown below. (Number Lock on the Numeric Key Pad must be OFF.)



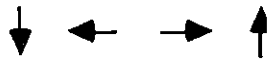
CRT MANIPULATION KEYS

CENTER - Numeric Key #0
Numeric Key 0 is used to move the data under the cursor, to the center of the CRT.

REPAINT - Numeric Key #1
Numeric key #1 is used to repaint the CRT screen without Expanding, Contracting or Moving the contents. Repaint is used to "fill in" blanks in the image at the CRT raster after parts/conductors have been deleted or moved. Repaint is also used after changing Text Size to present Text at the new size.

RESET - Numeric Key #7
Numeric key #7 is used to reset the CRT presentation at the center of the CRT (like key 0) and reset the Zoom to Expansion Level 1.

PAN



These keys are used to move or pan the presentation at the CRT around to different locations of the CRT. In the process, it does not enlarge or contract the scale. The algorithm functions as follows: "The database content will have its X and Y origin incremented by an amount determined by the Expansion Level and in a direction established by the Pan arrow selection located at the edge of the CRT at the time a Pan key is struck, such that it is centered at the CRT. It will not move the CRT presentation outside the upper right quadrant of the Cartesian Coordinate System.

To PAN, press Numeric keys 2, 4, 6 or 8, depending upon the direction you wish to pan. The scale will remain unchanged but the feature that was at the side of the CRT will be repositioned to the center of the CRT.

An alternate method of "Panning" is "Auto Pan." Simply place the cursor on the extreme edge of the CRT (top, bottom, right, left) and the presentation will be "panned" in a direction opposite to the side where the cursor is located.

EXPAND - Numeric Key #9 (PG UP)
This key is used to expand or double the scale (decrease the area) of presentation at the CRT by a factor of 2, and centered on the current location of the cursor. To use Expand, place the cursor at the area to be expanded. Press Numeric key button #9 and the item at the cursor location will be moved to the center of the CRT and the

presentation will be doubled in scale. There are eight expansion levels available.

CONTRACT - Numeric Key #3 (PG DN)
This key is used to contract or reduce the presentation at the CRT by one half, centered on the current location of the cursor. To use Contract, place the cursor at the area to be contracted. Press Numeric key button #3 and the scale will be reduced in size by one-half (more data presented at CRT).

WINDOW ZOOM - Numeric Key "." (DEL)
This key is used to define a "box" around an area which, after being defined, is automatically expanded to fill the CRT.

To create a "Box," select the DEL Key. The Command Menu will be replaced with:

1 START
2
0 EXIT

Position the cursor over any one of the four corners of the Box and select F1 START. A rectangle will be originated at the CRT and the command menu will be changed to:

1 COMPLETE
2
0 EXIT

Pull the mouse in the correct direction to form the correct box. When the desired area is "boxed," select F1 COMPLETE. The area enclosed in the Box will be expanded to fill the CRT.

Stop CRT Repaint - ESC Key
The ESCAPE key is used to stop the CRT Repaint during a long repaint. Use Numeric Key 1 to complete the repaint.

Chapter 2

THE IN/OUT ROUTINE

2.0 INTRODUCTION

The IN/OUT Routine is the first of the nine major or top level routines. It has a number of subroutines, all associated with Entering or Outputting the contents of the Schematic or Logic Diagram database.

When F1 IN/OUT of the top or major routine is selected, the top level routine menu is replaced with the subroutine of IN/OUT as shown below:

1 JOB IN
2 JOB OUT
3 ASCII IN
4 ASCII OUT
5 SHEETS
6 ECO
7 REPORTS
8 PLOT
9 NEW SCH
0 EXIT

2.1 BINARY JOB VS. ASCII FORMATS

The I/O routine handles two formats of Inputs and Outputs: BINARY and ASCII.

This Binary format is unreadable by the designer but can be read very quickly by the Logic Program. Binary files are identified by their file name and the extension .SCH, thus: XXX. SCH.

"ASCII," on the other hand, is designer readable information. The ASCII format can be brought up in a Text Editor and viewed as lines of data, and can be changed with the Text Editor. ASCII files are identified by the file name followed by the extension .TXT, thus: XXX.TXT.

The format of a ***PADS-LOGIC*** ASCII file is available from CAD Software, Inc., and with this knowledge, users of the ***PADS-LOGIC*** Program have the advantage of "open architectural software" in that they can interface systems both "to" and "from" the ***PADS-LOGIC*** database.

All files, both Binary and ASCII are stored by default in a subdirectory called \PADS\LOGFIL. It can be accessed via the path CD\PADS\LOGFIL. When in LOGFIL, typing DIR results in a listing of all files contained in LOGFIL.

2.2 JOB IN SUBROUTINE

The JOB IN subroutine recalls a Schematic previously stored on the hard disk, by the JOB OUT subroutine, and places the file in memory and displays it on the CRT. It is accessed by (1) function button F1 or (2) pressing Mouse button #1 after positioning the highlight bar over F1 JOB IN. The communications window located at the bottom of the CRT asks the question:

JOB INPUT FILE NAME (CR=CAE.SCH):

Respond by typing in the name of the binary job file that is to be recalled, followed by Carriage Return. The extension .SCH is not required. (To recall the Backup or Security file, simply strike the Carriage Return key.) Alternately respond: * <CR> and a pop-up window will appear at the CRT. This pop-up window contains an alphabetical listing of files in the LOGFIL directory. Use the mouse to position the cursor over the name of the job file to input and press mouse button #1. The named file will be inputted. NOTE: A long listing of files may be cycled through the pop-up window by selecting MORE on the pop-up window with the mouse cursor.

The message "WORKING..." is displayed, and in a few seconds, the design is moved into memory and displayed at the CRT. In using the JOB IN subroutine, be aware of the following:

(1) If the job file requested is not on the disk, an error message:

FILE NOT FOUND \PADS\LOGFIL\XXX.SCH

will be displayed in the communications window.

(2) If a design is currently in memory and displayed at the CRT, it will be overwritten by the new Binary .SCH file recalled from the hard disk.

(3) Design data in a Binary format cannot be added to existing Binary design data currently in memory.

2.3 JOB OUT SUBROUTINE

The JOB OUT subroutine is used to store the design data onto the hard disk in a Binary format. After a design is stored on the hard disk, it also remains in memory and is displayed at the CRT after its file has been written onto the hard disk.

The JOB OUT subroutine is activated by (1) pressing Function Key F2 or (2) pressing the appropriate mouse key. To the message:

SCHEMATIC OUTPUT FILE NAME (CR=CAE.SCH):

type in the desired name of the output file followed by Carriage Return and the file will be written to the hard disk. The CPU message window displays "WORKING..." for a few seconds, indicating that the job is being stored to the disk. Make a practice of watching for this message immediately after striking <CR>. This is your assurance that your design is being stored.

The name of the design file can be 1 to 8 characters in length. Use all Alpha and Numeric characters, as well as the following special characters: !, @, #, $, &, _, {,%, }, ~, ` we recommend, however, that special characters not be used unless absolutely necessary.

NOTE: A space or a tab is NOT accepted in the Job Name.

To store the design as a "Backup" file, simply use the Carriage Return key. The prompt:

OVERWRITE FILE Y/N?

may appear in the communications window. If you wish to

overwrite the existing stored Backup job file, respond with "Y"; the stored Backup design will be destroyed by being overwritten with the current design. If you do not want to overwrite the existing job, respond with "N" and provide a different name to store the job.

2.3.1 BACKUP OR SECURITY FILE

You will be relieved to know that if you have inadvertently made an error in storing jobs, the ***PADS-LOGIC*** Program has a unique data security feature called "Backup" that allows you to take corrective action if you discover your error in a timely fashion.

The Backup subroutine automatically acts to make a Backup file of the database in memory at intervals of time selected by the designer. When this occurs, the CPU message center window displays the word:

"WORKING..."

during the few seconds the backup file is being placed on hard disk. When the backup file is made, it overwrites the previously made backup file on disk.

The purpose of the Backup file is to give you a convenient way of recovering your work investment, in the event of an emergency such as a power failure, hardware failure, a software bug, or corruption of your Job in the Job Files. The Backup file is a Binary .SCH file maintained in the LOGFIL subdirectory under the name CAE.SCH.

NOTE: This backup file is not to be confused with the backup file created by the DOS Editor or any other editors you may have on your system. It can be identified by its extension ".SCH".

CHANGING THE BACKUP FILE INTERVAL

You may change the Backup Interval using the Routine of SETUP and PARAMETERS. Set the Interval (in minutes) to between 1 minute and 30 minutes. Note that our experience has shown that taking a backup file every 5 minutes is an optimum time for minimizing the loss of data. We, therefore, recommend that you leave the job interval time set at 5 minutes.

You will recall from Section 2.3.1 above, that another way of making or "forcing" a Backup file is to select Job Out of the In/Out routine, and then strike the Carriage Return. This will create a Backup file, overwriting the previous Backup file. To recall the Backup file into memory, simply use the In/Out routine, select JOB IN (key 1); the message:

JOB INPUT FILE NAME (CR=CAE.SCH):

will appear. Strike the Carriage Return, and the current Backup Job file will be read into memory, overwriting anything that is currently in memory.

Let's now describe a scenario of how the Backup file can be used. Let's say that you have been working on a large schematic for six or eight hours and have not saved any interim Job Out files for the past several hours--a bad procedure indeed!! Let's further imagine that the power cord of your Personal Computer is inadvertently pulled. What do you do? What recovery action do you have? If you power up your Personal Computer and call up the last file created using JOB OUT, you will find that it may be six or eight hours old if you have neglected to practice saving interim job files, and thus you may have lost six to eight hours of your work effort.

Here's how you save your work: As soon as you recognize the Job File just brought back in is several hours old, select JOB IN and hit Carriage Return. This will bring up the last Backup file (a "good" file) that was created just before the power cord was inadvertently pulled, which would be at the most, 5 minutes old. SPECIAL WARNING: CALL UP THE BACKUP FILE IMMEDIATELY, before it, in turn is overwritten with the old, out of date job file you brought into memory! You have 5 minutes or whatever Backup interval is set, after which the "good" Backup file on disk is overwritten with the out of date job you have in memory.

The concept of a Backup or Security file is a very powerful and useful security recovery technique. However, when using it, you have to be alert as to the timeliness with which you take remedial action.

CRASH RECOVERY: There may be times, due to a "glitch," or a hardware failure, your system will either Crash or Lockup. No problem! Simply reboot the system and call up the BACKUP.JOB (CAE.SCH) file, which at most will be only a few minutes out of date, and proceed with the design.

2.3.2 NOTES ON .SCH FILES

1. A file created with the JOB OUT subroutine will have the extension .SCH automatically assigned to the file name.

2. The nomenclature "Job" in the context of In/Out refers to Schematic Job files in Binary format. This information may include the Sheet Outline, parts, connections, and all other data contained in the Schematic database.

3. Caution! You must be aware of the difference of "Storing" or Outputting, versus "Recalling" or Inputting. The danger is losing the contents of a job in which you have invested many hours of your labor. REMEMBER THE FOLLOWING: JOB IN means to Bring IN stored information FROM the hard disk to the CRT and memory. JOB OUT means to move data from the memory and CRT, OUT to the hard disk. The classical error is for an Engineer to work on a Schematic for several hours and at the end of the shift, store the job (he thinks). In the process, instead of selecting the JOB OUT subroutine, the engineer inadvertently selects JOB IN. The unfortunate results are recognized on the next shift when the response:

FILE NOT FOUND

is given when trying to Recall the schematic that was thought to be stored.

4. A Suggestion: When intending to store a job, think "I want to move and store the file from memory and CRT, OUT to the disk," therefore, use JOB OUT. Secondly, when wanting to recall a schematic, think "I want to recall from disk and load IN memory," therefore, use JOB IN.

When making a JOB OUT, closely observe that the CPU message window shows "WORKING..." for a few seconds. If you want to

be VERY sure that you have stored your design, before you power down the CPU, simply recall the schematic that you have just stored using the JOB IN subroutine. If it "Recalls," then it has been stored properly.

2.4 ASCII IN SUBROUTINE

The ASCII IN subroutine is used to move an ASCII file from the hard disk into memory, to convert the ASCII file into a Binary format, and to display the schematic at the CRT. The ASCII IN subroutine is activated by the selection of the F3 Function Key (ASCII IN), or the appropriate mouse key. Respond to the message:

ASCII INPUT FILE NAME (CR=STARTUP.TXT):

with the name of the ASCII file, followed by a Carriage Return.

When utilizing this subroutine, be aware of the following:

(1) An ASCII file is automatically converted to a Binary format when recalled into memory.

(2) An ASCII file is added to an existing Binary file already in memory and displayed at the CRT.

(3) In the process of reading in an ASCII file, data checks are made on the contents of the file to eliminate, where possible, data errors before the conversion to a Binary job file occurs. This eliminates the tedious process of correcting input data errors. In most instances, an input data line with an error will not be converted into Binary until the offending error is corrected.

Any input file errors can be sent to a file for later analysis by responding to the request for the error file name message with a file name. Alternately, any errors can be sent directly to the printer by responding to the Error Message requesting a file name, by striking the Enter <CR> Key. We suggest the preferred procedure is to send the error file directly to the printer.

(4) A long (say, 200 IC schematic) ASCII file will take several

minutes to be read in. During read in, "WORKING..." is displayed in the CRT Window.

At this point you are probably wondering "Where does an ASCII Input file come from?, How is it created?, How do I use it?" These are all good questions and are answered in the next section.

2.5 ASCII OUT SUBROUTINE

This subroutine is used to convert an existing schematic in memory into an ASCII file and to store this file on hard disk or print it at the Matrix printer. To implement the ASCII OUT subroutine, select the F4 Function Key; a Window display, as shown below, appears at the CRT.

SCH	CAE	TEXT
LINES	DECAL	PTYPE
PART	CONN	ALL

ASCII OUT FILE CATEGORIES

The selection of an ASCII OUT category with the mouse cursor, will result in the following being contained in the ASCII file:

SCH — A listing of general parameters and Set Up values, such as Grid, etc.

CAE — A listing of general parameters of the Drawing Sheet being used.

TEXT — A listing that includes Free Text Strings, together with location, level, size, etc.

LINES — A listing of all Lines, including Sheet Outline and Part Decal Outlines, together with location, level and width.

DECAL — A listing of Part Decals, their content, together with their location.

PTYPE — A listing of the packaging type data associated with the Parts on the schematic.

PART — A listing of the Parts on the schematic including their Reference Designations.

CONN — A list of the Connections on the schematic.

ALL — A list of the entire Schematic including all the above items on all sheets.

NOTE: An ASCII OUT file intended to be read back into the Program via ASCII IN and used to restore or recreate the complete database for a board, must use the ALL choice.

Using the cursor, select the contents of the design that you wish converted into an ASCII file; then select F2 GO. In response to the message:

ASCII OUTPUT FILE NAME (CR=PRINTER):

enter the name by which you wish the ASCII file to be stored. Use of the Enter key will result in a printout on the printer, and a file will not be created.

The same rules for naming a JOB OUT file apply to naming of the ASCII OUT file. The extension .TXT is automatically assigned to an ASCII stored file.

ASCII OUT files can be used in the following manner:

(1) You can obtain a printout of the contents of the ASCII file and examine it in greater detail than when this file is graphically displayed at the CRT.

(2) Certain changes can be better implemented using the text editor, PEDIT and editing the ASCII file, and then reading the edited ASCII file back in, as opposed to graphically implementing the same change at the CRT.

(3) The ASCII file can be used to interface a file created in the

PADS-LOGIC Program with other programs or activities. (A PADS-to-Other ASCII conversion will be required.)

(4) The ASCII Out/In feature is an excellent database diagnostic tool. Should database items ever become contaminated as a result of hardware or software bugs, make an ASCII Out file of the suspected file, erase the contaminated file from memory, and read the ASCII file into memory using the ASCII IN feature. The ASCII In Data Checking program will usually identify the contamination, allowing you to delete it and replace with good data.

2.6 SHEETS

The subroutine of SHEETS is used for four functions as follows.

(1) to Select and Display a sheet of a multi-sheet schematic,
(2) to Add an additional sheet to a schematic,
(3) to Change the number of a previously numbered sheet, and
(4) to Delete a sheet from a multi-sheet drawing. (All Data must be removed from the sheet before deletion.)

When the SHEETS subroutine is selected, the Command Menu changes to the following:

IN/OUT
SHEETS
1 SEL SHEET
2 ADD SHEET
3 REN SHEET
4 DEL SHEET
8 HIER IN
9 HIER OUT
0 EXIT

2.6.1 SEL(ECT) SHEET

The subroutine of SELECT SHEET is used to Select and display a selected sheet of a multi-sheet schematic. When selected, the following prompt appears:

**CURRENT SHEET IS X -
ENTER NEW SHEET NAME/NUMBER>**

In response, enter the new sheet number or name, in the case of a hierarchical sheet <CR>. The selected sheet will then be displayed at the CRT.

SEL SHEET can be used only with a multi-sheet schematic and on valid sheet numbers or names. The error message of:

BAD SHEET NUMBER

will be displayed if an invalid sheet number is selected.

There is a Modeless command SHX which allows you to change to Sheet X at any time. X can be a sheet number or name.

2.6.2 ADD SHEET

F2 ADD SHEET is used to add new sheets to a schematic. Up to 256 sheets, depending upon available memory and contents of each sheet, may be contained in a schematic.

To add a sheet, select F2 ADD SHEET. Respond to the prompt:

OK TO ADD SHEET [X] (Y/N)>

Where X is the next sequential sheet number to be added to the schematic. A "Y" will result in a new blank sheet being displayed at the CRT. The sheet size will be the same as the proceeding sheet. Finally, the purple window will indicate:

SHEET X/Y

Where X is the new sheet number and Y is the total number of sheets.

A Sheet Outline is brought in from the library whenever a new sheet is added. The Default Sheet size is D, with sizes A-D as well as A4-A0 (International) also available. The SETUP, PARAMETERS subroutine is used to change the Default Sheet Size.

2.6.3 REN(UMBER) SHEET

F3 REN SHEET is used to renumber an existing sheet that is contained in the schematic.

To Renumber a sheet, select F3 REN SHEET. Respond to the prompt:

ENTER SHEET NUMBER TO RENUMBER>

with the number of the sheet to be Renumbered <CR>. Respond to the prompt:

RENUMBER SHEET X TO>

with the new sheet number <CR>

NOTE 1: The new sheet number must already exist in the schematic, otherwise the error message:

BAD SHEET NUMBER

will be created.

NOTE 2: When a sheet is Renumbered, the numbers of the sheets higher than the new number will be adjusted to accommodate the new number. Example: a 10 sheet schematic - Sheet #10 is renumbered to Sheet #8. The results: old sheet #8 becomes new sheet #9, old sheet #9 becomes new sheet #10.

2.6.4 DEL(ETE) SHEET

F4 DEL SHEET is used to delete a sheet from sheets used to document a design. To use, select F4 DEL SHEET and respond to the prompt:

ENTER SHEET NUMBER TO DELETE>

with the number of the sheet to delete <CR>. After confirming the deletion, the sheet will be deleted if in Design and NOT the current sheet displayed at the CRT. If the deleted sheet is the current sheet,

an error message will be produced. A Sheet that contains Electrical data cannot be deleted. An attempt to delete will produce an error message.

NOTE: The Number of the sheets higher than the deleted sheet will be adjusted to account for the deleted sheet. Example: A four-sheet schematic - Sheet #1 is deleted. The results: old sheet #2 becomes new sheet #1, old sheet #3 becomes sheet #2, etc.

2.7 ENGINEERING CHANGE ORDER ROUTINE (ECO)

OVERVIEW

ECO is the routine in which changes are made in the schematic that is used to update the Board database and, secondly, ECO accepts changes originating in the Board database and uses the file to automatically update the schematic.

ECO

Be in the IN/OUT top level Menu and select F6 ECO. When ECO is selected, the menu changes to as shown below:

IN/OUT
ECO
1 TO PCB
2 FROM PCB
0 EXIT

F1 TO PCB is used to batch changes in the Schematic and create a file in the directory \PADS\LOGFIL that can be sent to the PCB database. If the PCB CAD software has a Schematic update feature, its database can be updated, using the change data contained in this file.

F2 FROM PCB accepts ECO change data from the PCB database and auotmatically updates the schematic.

NOTE: The format of the ECO Change Files is the ***PADS-PCB*** ECO ASCII format. Use this file with other PCB systems. Their

ECO files will have to be edited to conform to the ***PADS-PCB*** ECO format.

2.7.1 TO PCB

When F1 TO PCB is selected, the following prompt appears:

SCHEMATIC OR ASCII FILE TO COMPARE WITH>

Respond with the name of the file on hard disk that you wish to compare the schematic currently in memory to. This can be a LOGIC schematic or an ASCII file created from a ***PADS-PCB*** job.

Next, respond to the prompt:

OUTPUT ECO FILE NAME>

with the name you wish given to the different or ECO file. The extension .ECO will be automatically applied to the file name.

The system will compare the two files and create a difference listing, using as its name the ECO File Name given above.

This ECO file may be examined at the CRT using PEDIT, or it may be printed using the DOS PRINT command. A sample ECO made to a schematic is shown below:

```
*PADS-ECO*
*REMARK*    old file:  \PADS\LOGFIL\temp1.asc
*REMARK*    new file: \PADS\LOGFIL\temp0.asc
*REMARK* created by ECOGEN version of Nov 10 1989
            09:11:43
*DELPIN*
C2.2    GND
C2.1    VCC
C2 DCAP, 10UF@DCAP
*REMARK* Deleted pins: 2,  Added pins: 0
```

This report states the following: The schematic name TMP0.ASC differs from the schematic TMP1.ASC by the following:

1. Connection made to pin 2 of C2 called GND has been deleted.
2. Connection made to pin 1 of C2 called VCC has been deleted.
3. Part C2, DCAP, 10UF has been deleted.

The file resides in the LOGFIL subdirectory and is called up automatically by the ***PADS-PCB*** Board Software and used to update the board database. Non-***PADS-PCB*** Board CAD systems can also access and use the ECO file.

Items that are inlcuded in an ECO from the Schematic to the PCB database include:

1. Added Parts
2. Deleted Parts
3 Added Connections
3. Deleted Connections
4. Combining Nets
5. Renaming Nets
6. Renaming Parts
7. Changing Part Type of Parts

NOTE: Under certain conditions (i.e., one, two or three-pin parts), Renaming Parts can cause the TO ECO file generator to use DEL PIN, DEL PART, ADD PIN and ADD PART rather than RENAME PART. We suggest that Renaming of Parts be done in the PCB Job and the schematic be updated using FROM PCB.

2.7.2 FROM PCB

F2 FROM PCB is used to take an ECO File generated in the Board CAD system and automatically update the schematic.

To use, have the schematic to be updated in memory and displayed at the CRT.

Be in the ECO Routine and select F2 FROM PCB. Respond to the prompt:

INPUT PCB ECO FILE NAME (CR=ECO.ECO):

th the name of the ECO file created in the Board CAD system.

Respond to the prompt:

ERRORS OUTPUT FILE (CR = PRINTER)

with either <CR> is you wish any errors printed, or if you wish a file created, provide the Error File Name.

The current schematic in memory will then be updated with the ECO File.

Items that can be assigned back to the ECO File from the PC Board include:

1. Renamed REF DES
2. Swapped Pins
3. Swapped Gates

2.8 REPORTS

The REPORTS routine is used to provide seven different types of information on the schematic in memory. These seven types of information are:

(1) Net List
(2) Bill of Material
(3) Off Page List
(4) Part Status
(5) Net Status
(6) Unused Pins
(7) Job Limits

Reports can be directly outputted to the printer or created as a file to be printed at a later time.

To use the REPORTS routine, be in the IN/OUT Routine and select F7 REPORTS. When selected, the Menu window changes to as shown below:

REPORTS
1 NET LIST
2 BILL MAT

3 OFF PAGE LIST
4 PART STATUS
5 NET STATUS
6 UNUSED
7 JOB LIMITS
8
9
0 EXIT

2.8.1 NET LIST

The NET LIST subroutine is used to generate a Net List for interfacing to ***PADS-PCB***, P-CAD and other Board CAD Systems as well as Logic Simulation. When F1 NET LIST is selected, a CRT screen of "Outputs" is presented as follows:

PADS-PCB	FUTURENET	CADSTAR	PCAD
TANGO	MENTOR	CADNETIX	VIEWLOGIC
SUSIE	PSPICE	OTHER	

Some of these Net Lists may not be "Active," depending upon the configuration of your ***PADS-LOGIC*** software.

Select the desired Output with the cursor, and respond to the prompt:

NET LIST OUTPUT FILE NAME (CR=REPORT):

The file will be created in ASCII using the extension .ASC for **PADS-PCB**, .NET for FutureNet, etc., etc. and may be Edited with PEDIT or printed with DOS PRINT. Listed below are the **PADS-PCB** and FutureNet formats.

PADS-PCB:

When outputting to PADS-PCB, the file WIDTH.DAT is the PADS

directory is read. This file contains the Default Track Width to assign to Nets as well as the Net Names to assign special widths to, i.e., PWR, GND. This file is in ASCII and can be modified by the user.

The format of a **PADS-PCB** Net List is as follows:

```
*PADS-PCB* FILE INFORMATION
*REMARK* \PADS\LOGFIL\1.SCH --ALL SHEETS

*PART*          ITEMS
U1              74LS04@DIP14
U2              74LS04@DIP14
*NET*
*SIGNAL* $$$2104 13
U2.2 U1.11 U1.12
*SIGNAL* $$$2125 13
U2.1 U1.13 U1.2
*SIGNAL* $$$2137 13
U1.5 U1.4 U1.9
*END*       OF ASCII OUTPUT FILE
```

FUTURENET:

When outputting to Futurenet or any other Net List besides ***PADS-PCB***, the question is asked:

SHEET TO OUTPUT (<CR>=ALL SHEETS)

If only one sheet is to be processed, type the sheet number, otherwise Carriage Return.

When outputting to Futurenet or any other Net List besides ***PADS-PCB***, the file FUTURE.DAT is also read. This file coordinates the matching of ***PADS-LOGIC*** attribute names to Futurenet attribure names, i.e.,

```
PART DESC:    80
PART DECAL:   81
```

This enables you to export the ***PADS-LOGIC*** attribute information

to Futurenet. Any ***PADS-LOGIC*** attribute not listed in the FUTURE.DAT file but defined in your schematic will cause the routine to query you for the Futurenet attribute number:

ATTRIBUTE NUMBER FOR [MFG #1:]>

Be sure not to use values 1-10 as these are predefined by Futurenet.

The format for the same Net List as above, but in FutureNet format is as follows:

```
NETLIST, 2
(DRAWING,\PADS\LOGFIL\1.SCH
DATA,53,ALL SHEETS
DATA,54, Tue Dec 19 11:45:22 1989
)
(SYM,1-1,0
DATA,2,U1A
DATA,3,74LS04
DATA,83,HEX INV
DATA,84,
DATA,82,74HCT04

DATA,85,TEXAS INSTRUMENTS, INC.
DATA,86,
DATA,80,DIP14
DATA,23,1
DATA,21,2
)
(SYM,1-1,1
DATA,2,U1B
DATA,3,74LS04
DATA,83,HEX INV
DATA,84,
DATA,82,74HCT04
DATA,85,TEXAS INSTRUMENTS, INC.
DATA,86,
DATA,80,DIP14
DATA,23,3
DATA,21,4
)
```

```
(SYM,1-1,2
DATA,2,U1C
DATA,3,74LS04
DATA,83,HEX INV
DATA,84,
DATA,82,74HCT04
DATA,85,TEXAS INSTRUMENTS, INC.
DATA,86,
DATA,80,DIP14
DATA,23,5
DATA,21,6
)
(SYM,1-1,3
DATA,2,U1D
DATA,3,74LS04
DATA,83,HEX INV
DATA,84,
DATA,82,74HCT04
DATA,85,TEXAS INSTRUMENTS, INC.
DATA,86,
DATA,80,DIP14
DATA,23,5
DATA,21,6
)
(SYM,1-1,3
DATA,2,U1D
DATA,3,74LS04
DATA,83,HEX INV
DATA,84,
DATA,82,74HCT04
DATA,85,TEXAS INSTRUMENTS, INC.
DATA,86,
DATA,80,DIP14
DATA,23,9
DATA,21,8
)
(SYM,1-1,4
DATA,2,U1E
DATA,3,74LS04
DATA,83,HEX INV
DATA,84,
```

```
DATA,82,74HCT04
DATA,85,TEXAS INSTRUMENTS, INC.
DATA,86,
DATA,80,DIP14
DATA,23,11
DATA,21,10
)
(SYM,1-1,5
DATA,2,U1F
DATA,3,74LS04
DATA,83,HEX INV
DATA,84,
DATA,82,74HCT04
DATA,85,TEXAS INSTRUMENTS, INC.
DATA,86,
DATA,80,DIP14
DATA,23,13
DATA,21,12
)
(SYM,1-1,6
DATA,2,U2A
DATA,3,74LS04
DATA,83,HEX INV
DATA,84,
DATA,82,74HCT04
DATA,85,TEXAS INSTRUMENTS, INC.
DATA,86,
DATA,80,DIP14
DATA,23,1
DATA,21,2
)
(SIG,,$$$2104,,5,$$$2104
PIN,1-1,6,U2A,21,2
PIN,1-1,4,U1E,23,11
PIN,1-1,5,U1F,21,12
)
(SIG,,$$$2125,,5,$$$2125
PIN,1-1,6,U2A,23,1
PIN,1-1,5,U1F,23,13
PIN,1-1,0,U1A,21,2
)
```

```
(SIG,,$$$2137,,5,$$$2137
PIN,1-1,2,U1C,23,5
PIN,1-1,1,U1B,21,4
PIN,1-1,3,U1D,23,9
)
```

2.8.2 BILL OF MATERIAL

The Bill of Material (BOM) Reports provides a listing of the Parts contained in the schematic currently in memory. When selected, respond to the prompt:

BILL OF MATERIALS OUTPUT FILE NAME (CR = PRINTER):

The following prompt requests the name of a Bill of Materials Format file which defines the format of the report. The default format file \PADS\BOMREP.DAT is supplied with the software. If you have generated an alternate format file, the default directory is \PADS\LOGFIL and the default file name extension is .BMF.

BOM FORMAT FILE (.BMF) (CR = \PADS\BOMREP.DAT)

The Bill of Materials routine will report on any part attribute that has been assigned to a part in the schematic. Examples of part attributes are "Description," "Manufacturer," "Value," etc. The user may specify up to ten attributes to be listed for each part. The attributes are listed in columns and the user has complete control over the order in which the attributes are listed, plus the starting and ending positions of each column. Line lengths of up to 200 characters are supported. If the length of an attribute exceeds the width of a column, then the software will automatically "word wrap" on to multiple lines within the space defined for that column.

Parts are normally sorted by Part Type name (which is the default) but the user may sort by Reference Designator or any other attribute selected for inclusion in the Bill of Materials report. The Part Type used can be just the name of the Part Type as it is used in the Parts Library, or it may be a combined name consisting of Part Type

name, Value and Tolerance. For example, the 1/4 watt resistor could have a Part Type name of R1/4W or R1/4W.4.7K,+/-5%. If the combined Part Type, Value and Tolerance option is selected, then each 1/4 watt resistor that has either a different Value or Tolerance rating, would be considered as a different part type.

To reduce the size of the report, parts that have identical attribute values and differ only by their Reference Designators will be combined into one entry in the report. For example, if U1 through U10, U12, and U16 to U18 are all identical parts, then one entry is generated with the Reference Designator column containing the string: U1-10 U12 U16-18

When entries are combined like this, the number of parts combined can be listed as a "Quantity" attribute. In the above example, the Quantity would be 14.

The layout of the Bill of Materials Report is controlled by a "form" or "dialogue box" shown in the following figure:

Bill of Materials Format Options
Report Title (%J = jobname, %D = date & time):

Bill of Materials Report for %J on %D

Part Attribute	Field Header	Start	End
QTY	Qty	1	5
REF-DES	Reference	6	15
PART TYPE	Part Name	16	30
MFG. #1	Manufacturer	31	45
$	Cost	46	50
PART DESC	Description	51	80

Attribute to sort:

Combine Value & Tolerance in Part Type (Y/N): Y
Separate Lines for each Ref-Designator (Y/N): N
Print Field of Column separators (Y/N): Y

Report Title: The user may specify his own title text for the report. The schematic job file name and date and time can be included in the title by inserting "%J" where the job name is to be inserted and "%D" where the date and time is to be inserted.

Attribute Name: Up to ten attribute names may be listed. This name is the exact name of the attribute as used in the Part Library or in Edit Text. There are three pre-defined special attribute names that

can be used:

QTY	to select the Quantity
REF DES	to select the Reference Designator
PART-TYPE	to select the Part Type name

NOTE: The order in which the attributes are listed on the form is not important, it is the starting column position that determines the order in which the attribute columns are listed in the report.

Field Header: Since the exact attribute name is not always what is wanted as a column header in the report, the user may specify the text of the column heading for each attribute.

Start: This is the starting character position of the attribute column on the printer or in the output file.

End: This is the ending character position of the attribute column. If the start position is "1" and the end position is "10" then the actual useable width of the column for the attribute data is only 9 characters, since a space has to be reserved to separate one column from the next.

Attribute to Sort: This is the name of the attribute by which the report is sorted. If left blank, the parts will be sorted by Part Type name.

Combine Value and Tolerance in Part Type (Y/N): This controls whether the Value and Tolerance attributes of a part are combined into the Part Type name.

Print Row of Column separators(Y/N): Select whether printable ASCII characters are used to separate the rows of columns to give the report a more "tabular" appearance.

After making any required changes to the report format, select F9 to continue or F0 to Exit out of the Bill of Materials routine.

If you have made changes to the report format, the following prompt will be displayed:

SAVE NEW BOM FORMAT (Y/N)?

Select "Y" if you wish to save the new format in a file, select "N" if your changes are just temporary for running this report only. If you select "Y," the following prompt is displayed:

BOM FORMAT FILE (.BMF)
(CR = \PADS\BOMREP.DAT)>

The default file name is the same file as you selected prior to the form being displayed. If you specify a file name, the default extension is .BMF and the default directory in which it will be stored is \PADS\LOGFIL.

The following is a format of a BOM used. The BOM Default values are unchanged.

Bill Of Materials for 1.SCH on Tue Dec 19 12:15:00 1989

Qty	Reference	Part Name	Manufacturer	Cost	Description
2	U1-2	74LS04	TEXAS INSTRUMENTS, INC.	$0.51	HEX INV

2.8.3 OFF-PAGE LIST

This report provides the location Sheet Number and X,Y coordinates of all Off-Page Symbols, GND Symbols and Power Symbols contained in the schematic.

When selected, respond to the prompt:

OFF-PAGE LIST OUTPUT FILE NAME
(CR = PRINTER):

with the name of the output file you wish to create for the report.

A "Potential Error" message is given when only One Off-Page Reference is contained in a Net.

HINT: To quickly locate an Off-Page symbol at the CRT, using the Report as a reference, proceed as follows:
Using Modeless macro commands:

1. Use SHX, where X is the Sheet Number
2. then SXXXX YYYY where XXX is the X coordinate and YYY is the Y coordinate of the Off-Page Symbol as listed on the report. The correct sheet will be found and the cursor will be located over the symbol.

The format of the OFF PAGE Report is as follows:

OFF PAGE REFERENCE REPORT -- \PADS\LOGFIL\DEMO.SCH --
Thu Dec 21 09:03:32 1989

11
SHT 1 X5900 Y7300 SHT 1 X11300 Y4000

13
SHT 1 X8000 Y4200 SHT 1 X12800 Y7800

16
SHT 1 X12700 Y3100 SHT 1 X8500 Y4800

21
SHT 1 X11300 Y3800
Only one off page reference in net.

39
SHT 2 X9300 Y5200 SHT2 X4200 Y5300

40
SHT 2 X12300 Y6900 SHT 2 X10700 Y5700
SHT 2 X4600 Y6900

GND
SHT 2 X5000 Y3300 SHT 3 X5400 Y1500
SHT 2 X5000 Y4700 SHT 4 X12300 Y5200
SHT 4 X12300 Y3400 SHT 4 X12300 Y7000

VCC					
SHT 4	X12300	Y6300	SHT 3	X5400	Y2500
SHT 4	X12300	Y4500	SHT 4	X12300	Y8100
Z					
SHT 2	X10700	Y5900	SHT 1	X11300	Y4600
SHT 2	X4200	Y5100			

2.8.4 PART/NET STATUS

The PART STATUS (and NET STATUS) Reports are two of the most important reports produced. Their intended use is to present in an organized text format (rather than graphical), all pertinent information on the schematic, as well as to call out possible Logic conflicts.

NOTE: PART STATUS and NET STATUS are separate reports!

The intended use of the PART and NET STATUS Report is for the Design Engineer to, after the Logic is captured, but before spending time on final drawings, simulation, etc., to run a PART/NET STATUS report and study it for potential problems.

When selected, respond to the prompt:

PARTS LIST OUTPUT FILE NAME (CR=PRINTER):

or

NET ANALYSIS OUTPUT FILE NAME (CR=PRINTER):

with the name of the output file you wish to create for the report.

The format for the Part/Net Status Report is as follows:

1. PART STATUS - Lists each Part, its REF DES and Part Type. Lists each pin on each Part, lists its Type and attached Signal Name. The following is a Part Status format report:

```
PART STATUS REPORT -- \PADS\LOGFIL\1.SCH --
Tue Dec 19  12:16:11  1989

U1              74LS04
U1.2[S-1] $$$2125          U1.4[S-1] $$$2137      U1.5[L-1] $$$2137
U1.9[L-1] $$$2137          U1.11[l-1] $$$2104     U1.12[S-1] $$$2104


U2              74LS04
U2.1[L-1] $$$2125          U2.2[S-1] $$$2104
```

NET STATUS - lists each Net, its Sources, Load and Undefined Nodes. Lists each Node contained within the Nodes. The following is a Net Status format report.

```
NET STATUS REPORT -- \PADS\LOGFIL\1.SCH --
Tue Dec 19  12:16:24  1989

$$$2104        SOURCES = 2     LOADS = 1      UNDEFINED = 0
U1F.12[S-1]              U2A.2[S-1]             U1E.11[L-1]
--- Net has multiple sources -- Be sure they are or-tiable.

$$$2125        SOURCES = 1     LOADS = 2      UNDEFINED = 0
U1A.2[S-1]               U1F.13[L-1]            U2A.1[L-1]

$$$2137        SOURCES = 1     LOADS = 2      UNDEFINED = 0
U1B.4[S-1]               U1C.5[L-1]             U1D.9[L-1]
```

2.8.5 UNUSED REPORT

The UNUSED Report is in two parts:

(1) Unused Parts and Gates
(2) Unused Pins on Each Part

HINT: To find a pin that is unused, use the Modeless command of SHX.Y where SH is Search, X is the Ref Des and Y is Pin. The correct Page will be brought to the CRT and the cursor will be located over the Unused Pin.

The format for the "Unused" Report is as below:

UNUSED ITEMS REPORT -- \PADS\LOGFIL\DEMO.SCH --
Thu Dec 21 13:17:45 1989

UNUSED GATE LIST

74LS00
U9C U9D
74LS04

74LS20
U8B U5C
74LS32
U5B U5C
74S74

CON\26P\ED
P1B P1C P1G P1I P1L P1T P1X

UNUSED PINS LIST

6167
U12.9 U12.11
U13.9 U13.11
U14.9 U14.11
U15.9 U15.11
U16.9 U16.11
U17.9 U17.11
U18.9 U18.11
U19.9 U19.11
U20.9 U20.11
U21.11 U21.9

74LS04
U6.2 U6.3 U6.4 U6.6 U6.5

74LS139
U1.1 U1.2

2.8.6 JOB LIMITS REPORT

The JOB LIMITS Report is important in that it indicates the maximum number of various items (Parts, Nets, Pads) your system, with its current memory, is capable of handling in a Design, and then indicates the count of these items used at the time of running the report.

The Report is in two parts:

1. Part One is a count of items common to the entire schematic.
2. Part Two is a page-by-page count as applicable to that page.

WARNING: From time to time, while capturing your design, run a complete JOB LIMITS Report and check that you are not approaching the limit for any particular item. If you should ever have occasion to exceed the "Maximum No. of Items" number for any item, you will not be able to continue on that schematic. The solution will be to break up the design into two schematics, run Net Lists for each schematic, then merge the Net List using the Text Editor.

The format for the JOB LIMITS Report is as follows:

JOB LIMITS REPORT -- \PADS\LOGFIL\1.SCH --
Tue Dec 19 12:16:54 1989

ITEM TYPE	LOCATION	MEMORY ASSIGNED	MAX NO# ITEMS	ITEMS USED
Sheets	DOS Mem	7680	256	1
Sht Gate/Net Ref	DOS Mem	7000	7000	15
Parts	Expanded Mem	12500	1250	2
[illegible]Para. Ref	Expanded Mem	20000	10000	16
[illegible]ara. Names	Expanded Mem	32000	32000	112
Nets	Expanded Mem	20000	2000	5
Nets chars	Expanded Mem	20000	20000	0
Other	DOS Mem	768	768	768

----------NEXT SHEET NUMBER 1 ----------

ITEM TYPE	LOCATION	MEMORY ASSIGNED	MAX NO# ITEMS	ITEMS
Text Strings	Expanded Mem	40000	2000	0
Text Chars.	Expanded Mem	24000	24000	0
Drawings	Expanded Mem	54500	1238	7
Pieces	Expanded Mem	10000	2000	45
Corners	Expanded Mem	57500	9583	107
Arcs	Expanded Mem	7000	500	0
Part Decals	DOS Mem	7848	109	24
Terminals	DOS Mem	10682	763	8
Part Types	Expanded Mem	11400	300	1
Gates	Expanded Mem	6000	600	6
Gate Pins	Expanded Mem	21600	3600	12
Gate Sig Pins	Expanded Mem	7200	450	0
Pin Num Names	Expanded Mem	7000	1400	0
Pin Names	Expanded Mem	7000	7000	
Gates/Comps.	Expanded Mem	42000	500	7
Comp pins	Expanded Mem	22500	7500	14
Nets names	Expanded Mem	18000	2250	0
Tie Dots	DOS Mem	2000	250	3
Off Page Refs	DOS Mem	35598	1271	0
Buses	DOS Mem	1600	50	0
Connections	Expanded Mem	45000	1500	9
Route Segments	Expanded Mem	19500	4875	22
Other	DOS Mem	20	20	20

2.9 PLOT ROUTINE

The PLOT Routine is used to produce a hard copy of all the sheets in the schematic or just the sheet in memory and currently being displayed. Plotting can be done to most Matrix printers, wet ink plotters (Hewlett Packard, Houston Instruments, Inc.), and Gerber photoplotters.

Either a full sheet or that part of the sheet which is being displayed at the system CRT can be plotted.

2.9.1 USING THE PLOT ROUTINE

From the IN/OUT Routine command menu, select F8 PLOT. The menu changes to:

```
IN/OUT
PLOT
1 PRINTER
2 PEN PLOT
3 PHOTOPLOT
4 APERATURES
0 EXIT
```

The selection of the output device places you in the post processing stream of the device selected. The next CRT sequence is as shown below.

After selection of the output device type (F1, F2, F3), the following form is displayed:

Plot all numbered sheets (Y/N):	N		
Plot all Hierarchial sheets (Y/N) :	N		
If single sheet - Plot window (Y/N):	N		
Plot Scaling Ratio:	1	to	1
Plot Rotation in 90 degree increments:	0		
Center plot on plotting area (Y/N):	Y		
If not centered - select margins (Y/N):	Y		
Left or Right margin (L/R):	L	Offset:	0
Top or Bottom margin (T/B):	B	Offset:	0
If no margins - select origin offsets - X:	0	Y:	0
Plotter Pen Width:	16		
Plot Jobname (Y/N):	Y		
On-line Plot (Y/N):	Y		
If off-line plot - enter filename prefix:	Demo		

The following describes each of the lines in the figure above.

1. PLOT ALL NUMBERED SHEETS (Y/N?)
Answer Yes if you wish all the Numbered Sheets in the Design to be plotted sequentially starting with the lowest number and ending with the highest number.

2. PLOT ALL HIERARCHICAL SHEETS (Y/N?)
Hierarchical Sheets are those sheets created during the process of creating a Hierarchical Design, wherein a Hierarchical Sheet, rather than a Numbered Sheet of the design has been selected. In the event of a Hierarchical Design, containing both Numbered Sheets and Hierarchical Sheets, a response of Yes to both Line 1 and Line 2 will result in all the Numbered Sheets being plotted first, followed by all the Hierarchical Sheets.

HINT: If you have a circuit used X times but do not want to plot all X sheets, respond N.

3. IF SINGLE SHEET - PLOT WINDOW (Y/N?)
This entry allows the plotting of whatever is in the CRT Window at the time. For example, if you wish to have a fairly large blown up area of a section on the Schematic, Expand the area until it is visible at the CRT and answer No to Lines 1 and 2 and Yes to Line 3.

4. PLOT SCALING RATIO: 1 TO 1
Generates the SCALE of the plot by multiplying the first entry and dividing by the second entry. The default (1 X 1) results in a scale of 1:1. For a scale of 2:1 set multiplier to 2 and divisor to 1. For a scale of 0.5, set multiplier to 1 and divisor to 2.

5. PLOT ROTATION IN 90 DEG INCREMENTS: 0
This is used to rotate the plot in 90 degree increments. The entry of 90, 180, 270 or 0 are legitimate entries and rotate the plot counter-clockwise acccordingly.

6. CENTER PLOT ON PLOTTING AREA (Y/N): Y
Allows the movement of the center of the plot with regard to the center of the printer paper or plotting area. The default is Y for centering.

NOTE: The assumption is made that the printer or plotter is loaded with the largest sized paper for which the printer or plotter is designed, i.e., a D-size plotter such as the HP-7580 should have D sized paper loaded to obtain the desired results; also, a wide printer should have wide paper loaded. If the center of the plotting area is NOT selected, the next line "if not centered" is operative.

7. IF NOT CENTERED - SELECT MARGIN (Y/N): Y
Allows margins to be selected, by activating the following two queries i.e., LEFT/RIGHT and TOP/BOTTOM.

8. LEFT OR RIGHT MARGIN (L/R): L OFFSET: 0
TOP OR BOTTOM MARGIN (T/B): B OFFSET: 0
Permits margins to be set left side, right side, top or bottom according to the selections and the offset. Units of offset are 0.001". An entry of 1000 is a 1 inch offset.

9. IF NO MARGINS - SELECT ORIGIN OFFSET:
X-OFFSET: 0 Y-OFFSET: 0
If neither centered or margin options are selected, then this allows the user to specify the exact placement of the design origin with respect to the plotter origin. An entry of 1000,1000 will cause the plot to be plotted with the design origin 1 inch up and to the right of the plotter origin.

NOTE: Offsets must be calculated taking previous Transformation Options into account, i.e., Scaling and Rotation are applied before offsetting.

10. PLOTTER PEN WIDTH: 16
This is relevant only if Pen Plotter was selected on the Output Device Screen. Change this value to reflect the actual pen size used for the plot. If this is not specified correctly then solid filled items such as pads and tie dots may not be drawn correctly. The pen size is reflected in mils (.016), not millimeters.

11. PLOT JOBNAME (Y/N):Y
Selecting "Y" will cause the JobName, Time and Date to be plotted on the drawing. Selecting "N" will supress the output of the Job Name, Time and Date.

12. ON-LINE PLOT (Y/N):Y
This allows you to override the Default On-Line or Off-Line setting for the output device chosen during installation of the software. Selecting "Y" will cause the output to go directly to the output device (printer, pen plotter or photoplotter). Selecting "N" will cause the output to go to a disk file.

13. IF OFF-LINE PLOT - ENTER FILE NAME PREFIX
You may specify the first 5 characters of the output plot file. The software will assign the last 3 characters of the file name to correspond with the sheet number of the plot. The extension is determined by the output device selected, i.e., .MPR for Matrix Printer, .PEN for pen plotter, and .PHO for photoplotter. For example, the plot file for sheet 5 for a photoplotter would be: ABCDE005.PHO, where ABCDE is the user specified portion of the file name. The files are stored in the \PADS\LOGFIL subdirectory.

2.9.2 PHOTOPLOTTER

If a plot is being generated for the photoplotter, a report titled "Photo-plotter Apertures Report" will be printed. This lists the apertures used by the plot. If the plot requires aperture sizes that are not available in the "Aperture Table" (see Section 2.9.3 on Apertures), a list of the apertures used is printed followed by a list of the unavailable aperture sizes required. The system will then request that the user enter an aperture number to be used for each of the unavailable sizes as follows:

ENTER APERTURE NO. FOR WIDTH: N
HEIGHT/ID; SHAPE: X: LINE

These new aperture assignments are temporarily made for this plot only, and do not affect the "Aperture Table".. To make these temporary assignments you will need a print-out of your "Aperture Table", which you would have obtained last time you updated the Aperture Table in the Defaults section.

Suggested method of picking aperture assignments:

(a) Choose an unassigned aperture if available.

(b) If no unassigned apertures remain, then choose one that is not used for this plot.

(c) If no more unused apertures remain, choose a used one that has the same shape and is closest in size.

(d) If there are no suitable apertures available, the plot must be cancelled (use F10 Key) and the Line Widths in the design must be modified to use fewer different sizes.

2.9.3 APERTURE

F4 APERTURE permits the editing or setting up of the Apertures used to plot the schematic, if a photoplotter is used.

When F4 APERTURE is selected, a figure similar to the following is presented:

DCode	Width	Ht/ID	Shape
10	5	0	LINE
11	10	0	LINE
12	12	0	LINE
13	15	0	LINE
14	20	0	LINE
15	50	0	LINE
16	70	0	LINE
17	55	0	RND
18	62	0	RND
19	70	0	RND
70	100	0	RND
71	70	0	SQR
20	100	55	OVAL
21	55	100	OVAL
22	100	55	RECT
23	55	100	RECT
24	70	0	ANNL
25	70	0	THER
26	200	0	ODD
27	0	0	
28	0	0	
29	0	0	
72	0	0	
73	0	0	

APERTURE TABLE

The screen displays the current aperture selections for the job. There are a total of 24 aperture positions for a normal Gerber photoplotter. If any positions are left unassigned, then any apertures required that are not in the table are automatically inserted by the Gerber post processor into the unassigned positions. The apertures are numbered with Gerber's unusual numbering system, i.e., 10-19, 70, 71, 20-29, 72, 73. A Width of zero indicates an unassigned aperture.

Each assigned aperture position requires a DCode Width, Height, or Internal Diameter, and Shape to be entered. The following shapes

are recognized:

LINE - A round shaped aperture used for line drawing
SQRL - A square aperture used for drawing rectangular pads
RND - A round aperture used for flashing pads
SQR - A square aperture
OVAL - An oval shaped aperture for flashing oval pads
RECT - A rectangular shaped aperture for flashing rectangular pads
ANNL - An annular aperture
ODD - Any other shaped aperture

NOTE: Only LINE and SQRL apertures can be used to draw a line--all others are flashed.

For all apertures which are circular in nature, the Width indicates the aperture's diameter. The Height would therefore be set to zero.

After performing the required modifications to the Aperture Table, select COMPLETE to return back to the PLOT Routines menu.

2.10 NEW SCHEMATIC

The routine NEW SCHEMATIC provides a convenient method of clearing the current schematic from Memory and the display, and starting a new and different schematic.

To use, be in the routine of IN/OUT and select F9 NEW SCHEMATIC. Respond to the prompt:

MAKE SURE YOU SAVED SCHEMATIC TO DISK!
OK TO CREATE NEW SCHEMATIC Y/(N)?

as appropriate. An "N" will cancel the routine. A "Y" will erase the current schematic (therefore, if you will want to save the schematic for use at a later time, be sure to make a .SCH Job file before erasing).

The current schematic is erased from Memory and a new, blank Sheet #1 is displayed.

Chapter 3

ADD PART ROUTINE

3.1 ADD PART

The ADD PART routine is used to bring a part from the ***PADS-LOGIC*** library into memory, display and manipulate it.

To use, be in the top menu command structure and select F2 ADD PART. Respond to the prompt:

NAME OF PART TYPE FOR NEW PART>

with the name of the new part to be added, i.e., 74LS04 <CR>. The Library will be searched for the part, the word "Working" is displayed in the Yellow window, and the part will be brought in from the Library and attached to the cursor. At the same time, the menu changes to the following:

ADD PART
1 COMPLETE
2 ROTATE
3 MIRROR X
4 MIRROR Y
5 ALTERNATE*
6 NEXT TYPE**
0 EXIT

*Present if an Alternate Decal exists for the part.

**NEXT TYPE is used (when adding a part) in those instances wherein there are multiple type gates within the Part: an LM3900 for example. Let us say there were two types of gates--Gate A and Gate B. When the part is first used, Gate A will be brought up and added to the schematic. If you wish to continue using only Gate A on the schematic, then continue to ADD or COPY Gate A. If, on the other hand, you also wish to use Gate B, utilize F6 NEXT TYPE.

Move the cursor, with the part attached to the final location on the sheet. Rotate and Mirror if necessary, and use F1 COMPLETE to final locate the part.

If you wish to "Browse" the Library, to select the part, respond to the prompt requesting the part name with *. A Popup Window will be displayed listing all the Part Types in the Library. Select the part with the mouse cursor and a click of the left mouse button.

NOTE: See Section 3.3 on NOTES IN USING ADD PART.

When the part is brought onto the schematic, the Reference Designation is automatically assigned. If the part is a gated Integrated Circuit, the gate modifier is also automatically assigned, thus: U1A, along with its associated pins.

3.2 THE SUBROUTINES OF ADD PART

3.2.1 COMPLETE

This command is used when you have located the part at its desired location and its presentation (Rotate, Alternate, Mirror) is as desired. F1 COMPLETE will enter the Part Decal into the Schematic database and detach it from the cursor. After COMPLETE, another copy of the part is immediately available attached to the cursor (white). If not required, the part addition is canceled with a click of the Right mouse button. The SetUp defined color and Part Text will appear after F1 COMPLETE is accomplished.

3.2.2 ROTATE

F2 ROTATE is used to Rotate the Part that is attached to the cursor. When selected, it will rotate the part 90° in the clockwise direction, or 90° in a counter clockwise direction; and when complete, return the Menu highlight bar back to F1 COMPLETE.

3.2.3 MIRROR X

F4 MIRROR X is used to flip or mirror the part attached to the cursor, in the X direction; that is to say, "flip the part over a Vertical

Reference line."

3.2.4 MIRROR Y

F5 MIRROR Y is used to flip or mirror the part attached to the mouse in the Y direction; that is to say, "flip the part over a Horizontal Reference line."

3.2.5 ALTERNATE (If an Alternate Decal Exists)

F5 ALTERNATE provides the capability of changing the current symbol to an alternate representation, while the part is being located or placed on the schematic. Example: ALTERNATE is used to create the DeMorgan equivalent of the part. For example, a 7402 is normally representated by a NOR gate; the selection of ALTERNATE will display the DeMorgan Equivalent.

3.2.6 NEXT TYPE (If more than one Gate Type in Part)

When adding a part with more than one gate type in the part, the NEXT option allows you to select the next type of gate in the part. This lets you select the gate within the part with the Electrical properties you desire.

3.3 NOTES ON USING ADD PART

NOTE #1: When calling up a part from the Library; you may use a number of conventions:

(1) By Specific Name
Calling the part by its specific name, i.e., 7410, 74ALS10, 74H10, 54L10, 54LS10, 54S10, R1/4W, etc.

(2) By Wild Card and Modifier
74* <CR> When the Wild Card * is used, a pop-up window appears and ALL components whose first two digits are 74 in the Library, will be Listed. Use the mouse cursor to select the specific part you desire.

(3) By Wild Card and No Modifer
* <CR>. When the Wild Card * is used with No Modifier, all Parts in the Library are Listed in a pop-up window. This is an excellent way to "Browse" the Library. Use the cursor to select the desired part.

(4) \ Back Slash With Modifier
When the Back Slash character is used in conjunction with a Modifier, all parts in the Library using the Back Slash and the Modifier will be Listed in a pop-up window. Example: \10 will display all IC's using \10 in the Part Type, i.e. 74LS10, 74ALS10, 5410, etc. Use the cursor to select the desired part.

NOTE #2: When a part is added, the Reference Designation Name and Gate Modifier is automatically assigned.

The auto assignment of REF DES uses the following rules:

1. As an ungated part is added to any sheet in the design, its REF DES is incremented by one, i.e., R1, R2, R3, etc.

2. A part may be deleted and the REF DES for that part will be re-used by the Auto REF DES feature.

3. The REF DES used by a part may be manually edited or changed. The blank space resulting from the change will be used by the Auto REF DES feature.

4. The gate of a gated component will have both a Reference Designation (U1, U2, U3, etc.), and a gate modifier (A, B, C, etc.), automatically assigned at the time the gate is added to any sheet of the schematic. For example: U1A, U1B, etc.

5. A gate may be deleted, or the REF DES manually changed, and the Auto REF DES feature WILL reuse the unused REF DES, the next time a gate of the SAME PART TYPE is added to any sheet on the schematic.

6. A data file called PKGTYP.DAT contains a list of REF DES prefixes assigned to Part Types. For example:

RES R
CMOS U
TTL U

When added, Part Types of Type RES will automatically be given the RES prefix R. If a part is added whose type is not defined in the PKGTYP.DAT file, the program will ask for a Prefix before adding the part to the schematic.

Chapter 4

THE ADD CONNECTION ROUTINE

The ADD CONN Routine is used to add, edit and modify connections (but not buses) on schematic sheets.

To use, be in the top menu schematic, have the cursor positioned over the pad (or connection) to which the connection is to be started from, and Select F3 ADD CONN. A connection (white line) will originate at this point and will move with the cursor.

The originating pin and/or connection net name appears in the Yellow Window.

The menu changes to the following:

ADD CONN
1 CRN/FINISH
2 DEL CRN
3 ANGLE
4 OFF PAGE
5 GROUND
6 POWER
0 EXIT

A connection is a graphical representation of a current carrying line (it is not merely a 2-D Line as is used in some widely used schematic systems) between two points. That is to say, a Connection in ***PADS-LOGIC*** contains two attributes:

(1) the graphical 2-D Line, and
(2) the connectivity data for the connection.

NOTE: The words "connection" and "conductor" are used interchangeably in this manual.

A net consists of one or more connections. A net must carry a Net Name. The Net Name will be assigned by the system automatically, unless specifically named by the designer. If assigned by the

system, the format is $$$XXXX. When assigned by the designer, the name should consist of alphanumeric characters.

4.1 CRN/FINISH

The F1 CRN/FINISH subroutine is used to add a 90 degree corner when placing a connection, or to finish or terminate the connection. In both instances, F1 CRN/FINISH will result in the desired action. The Right mouse button (F0 EXIT) cancels the connection.

To Finish the connection, the cursor with the connection attached, must be within one grid unit of either another connection, the desired terminating pin or a Bus. When connecting TO another connection, the name of the resulting net is the name of the current connection being put into the Database, NOT the connection to which the connection is being connected, unless the connection being connected to was interactively named by the designer or a Bus connection. Designer assigned names and connections made to a Bus always take precedence over system assigned names.

4.2 DEL(ETE) CORNER

The DELETE CORNER subroutine is used to erase, or delete the LAST corner placed in the connection.

To use, select F2 DEL CORNER, and the last corner placed in the connection will be deleted, unless there is only one corner remaining in the connection.

4.3 ANGLE

A corner, as put in using the F1 CRN/FINISH subroutine, is always at 90°. Occasionally, you may want a connection segment at a diagonal, rather than horizontal, or vertical. To do this, after having put in a corner with F1 CRN/FINISH, then Select F4 Angle. Subsequent movement of the cursor can result in a Diagonal segment at any angle of choice. To change from "Corner at an Angle" to 90° corners, use F1 CRN/SELECT.

4.4 OFF PAGE

F4 OFF PAGE is used to terminate a connection on a page without tying it to a part, with the intention of picking up the connection on a different sheet or elsewhere on the page. The OFF PAGE symbol with identical connection Names are used for this purpose.

To use, bring the connection to the point you wish the Off-Page Symbol to be located and select F4 OFF PAGE. The Off-Page Symbol is attached to the cursor and the menu changes to:

1 COMPLETE
2 ROTATE
3 ALTERNATE
4 MIRROR X
5 MIRROR Y

NOTE: The ALTERNATE is used to graphically indicate an IN or OUT Symbol; there is no difference insofar as the connectivity data is concerned.

ROTATE and MIRROR as required and place with F1 COMPLETE.

Respond to the prompt:

NET NAME>

with the Net Name to be used.

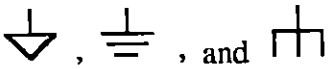
NOTE: You must remember to use the same Net Name at other Off Page locations of this Net. The use of the same Net Name will tie all usages into a common Net.

4.5 GROUND

F5 GROUND is used to terminate a connection with a Ground symbol. When so terminated, that connection is automatically assigned a net name that was specified when the symbol was created in the Library editor. A number of Ground symbols may be created with the Library editor, each with its own net name. The appropriate one can then be selected using the Alternate key. The

net name associated with the Ground symbol can be modified by using the EDIT NAME command and pointing at the segment to the Ground symbol.

Three symbols are available:



You may modify these and add additional symbols.

4.6 POWER

Power symbols are created and used in the same manner as Ground symbols. The name of the standard Power Net is +5V.

NOTE: The construction of the part in the ***PADS-LOGIC*** Library allows the including of standard Power (+5V) and Ground in the Part. A ***PADS-LOGIC*** Net List automatically includes a +5V and GND Net composed of these part pins. Thus, you need to add PWR/GND symbols on the schematic only for parts that do not include standard PWR/GND or for design practices such as "Ground all unused Gates," etc.

NOTES ON ADDING CONNECTIONS:

(1) Connections are always tied to a legal item (Pin, Bus, or other connections).

(2) Connections can start from Part Pins or other connections.

(3) Connections can finish on Pins, other connections, Buses, Off-page, Power and Ground symbols.

(4) Connections can NOT start from a bus.

(5) Connections can finish on a bus.

(6) Connections always have a Net Name assigned to them by the system. The Designer can overwrite this with the EDIT NAME routine.

(7) Connections are modified or moved, using the MOVE Routine.

Chapter 5

THE EDIT TEXT ROUTINE

5.0 INTRODUCTION

The EDIT TEXT routine is used to change the various types of text associated with a schematic or to edit FREE TEXT on the schematic. Specifically, Part Text items can be edited, as follows:

1. The Reference Designation Name can be edited to change the Reference Name or, if gated, to repackage the gate.

2. The Part Type can be edited, i.e., change 74LS00 to 74ALS00.

3. The Pins on the Part may be swapped interactively on the schematic at the CRT to facilitate the connection flow.

4. Connection Net Names and Bus names can be edited and added.

5. Tolerance, Part Value (ohms, etc.) as well as other parameters can be added and edited in parts.

6. The display of text associated with a part can be set On or Off.

7. Free Text can be edited.

5.1 EDIT TEXT

To use EDIT TEXT, be in the Top level Menu, place the cursor over the item to be edited (Pin Name, Ref Des or Part Type), etc., and select F4 EDIT TEXT.

EDIT TEXT is "intelligent" in that the text category is recognized, and the menu appropriate to the selected TEXT, will then be displayed for your subsequent action.

Example: If you have placed the cursor over the REF DES, the REF DES Name selected is presented on the prompt line, and a Red pop-up window with the REF DES highlighted in Blue, appears at

the CRT.

5.1.1 REF DES

To edit the REF DES, use the backspace key to erase the REF DES, then enter the new REF DES <CR>. In the process, observe the following:

1. A REF DES that duplicates a REF DES already on the schematic will not be accepted.

2. If the REF DES format is that used for a gate, i.e., REF DES stem and gate modifier, then this same format must be used.

5.1.2 PART TYPE

Select the PART TYPE Name with the cursor, the Part Type Name is displayed on the prompt line and the Red pop-up window appears with the Part Type highlighted in Blue. To edit the Part Type Name, use the back space key, then enter the New Part Type.

When a New Part Type is entered, the Menu changes to:

1 THIS GATE
2 THIS PART
3 ALL PARTS
0 EXIT

Select the Desired Action.

THIS GATE results in the Selected Gate being changed to the new part.

THIS PART results in this Part and its gates being changed to the new edited part.

ALL PARTS results in all parts of the same type that were edited, being changed to the new edited part.

NOTE: The ALL PARTS feature assumes a 74LS04 is a different part from a 74AS04 even though in the same part family.

The following rules are observed when editing Part Types:

1. A new part or gate with the same number of pins as the replaced part, will be replaced.

2. A part or gate with fewer pins than the replaced part or gate will NOT be replaced.

3. A part or gate with a greater number of pins than the replaced part or gate, will be replaced.

4. When changing Part Type to a New Part Type Name, the New Part Type must exist in the Library.

5. If the Part Type Name is the same as the current Part Type Name, then the Part Type in the schematic is replaced by the part type in the Library. Only the ALL PARTS option is available.

NOTE: The item selected for editing is changed to Blue on the Red Popup Window, and is displayed in the Prompt Message Line.

The Red pop-up window that appears with the selection of an item in EDIT TEXT is used to turn items ON or OFF from being displayed at the CRT, or from being plotted. Each of these items (Ref Des, Part Type or Pin Number), has an ON/OFF toggle that is toggled ON/OFF in the Red Window, by selecting the item with the cursor. ON will result in the item being displayed at the CRT and plotted; OFF suppresses the item at the CRT and during plotting.

5.1.3 PIN SWAP

The selection of a Pin Number will NOT result in the pop-up window, but will result in the following prompt:

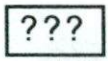
SELECT PIN TO SWAP WITH PIN XX.Y

Your next action is to select the pin on the gate that you wish to be swapped with XX.Y. When selected, the Pin Number and Name of the two selected pins will be swapped. The Pins must be on the component.

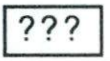
This Swap Pin feature provides a method of rearranging the pins on an individual part in the schematic, so as to make the flow of the schematic more pleasing. The Swap MUST occur before connections are made to the pins. Attempts to Swap Pins with connections will result in an Error Message.

5.1.4 PART TOLERANCE/VALUES

If the Part Electrical data section has been constructed using one or both of the Attributes VALUES: ??? or TOLERANCE: ???, a Box thus:



will appear with the part, for each Attribute. The VALUES: ??? may be changed to Values such as 10K, etc., and the TOLERANCE: ??? can be changed to ±5%, etc. To use, select EDIT TEXT, select the VALUE or TOLERANCE



and edit to the desired VALUE or TOLERANCE.

5.2 FREE TEXT

FREE TEXT on the schematic, using ADD TEXT, can be Edited with EDIT TEXT. Place the cursor over the Text to be edited and select F4 EDIT TEXT. Edit the Text and complete the Edit with Carriage Return <CR>.

5.3 NET NAMES

Nets are assigned Names at the time the connection is added to the schematic. The form is $$$NNNN, with NNNN being a non-repeated random function. These Net Names may be changed by placing the cursor over the Line representing the connections, and selecting F4 EDIT TEXT. The Net Name will appear on the prompt line. You may Edit the Name by backspacing and entering the New Name. You may move the Name box on the connection, containing the Net Name, using F2 MOVE.

NOTE: The first segment of the line representing the connection originating or terminating at a Symbol's pin, must be selected in

order to have the Net Name appear on the Prompt line.

5.4 **BUSES**

The "stem" portion of a Bus or the "Bit" portion may be changed using EDIT TEXT. The procedure is the same as used for editing a Connection Name.

Chapter 6

COPY ROUTINE

6.0 INTRODUCTION

The COPY Routine permits the duplication of Parts, 2-D Lines, Text, Hierarchical symbols, etc. The cursor selection is intelligent in that the system recognizes the item selected and presents the correct menu for the item selected.

6.1 PART

The COPY routine is used to copy or "duplicate" parts that are already available on a sheet of the schematic. (ADD PART, you will recall, is used to bring the part from the Library and place it on the schematic.) Once the part is available on a sheet in the schematic, COPY is used for subsequent duplication of the part.

To use COPY, be in the top level routines, place the cursor over a portion (Outline, Pin, Name, etc.) of the Part to be "copied" and select F5 COPY. The Part will be copied and attached to the cursor with the Outlines in White (or other user selected foreground color), No Text is displayed, and the menu will change to the following:

COPY
1 COMPLETE
2 ROTATE
3 MIRROR X
4 MIRROR Y
5 ALTERNATE*
6 NEXT TYPE
0 EXIT

*If Alternate Decal is present

The above subroutines function exactly as the same subroutines of F2 ADD PART, Section 3.1, and the description will not be repeated here.

6.2 NOTES FOR USING COPY

1. Note that the Part Reference Designation and Gate Modifier are automatically assigned at the time the COPY command is completed with the COMPLETE routine.

2. Note that Duplicate Ref Des Names are not possible.

3. Note that blank REF DES, created by Editing or Deleting a component, are reused by the COPY Part Routine.

6.3 TEXT

COPY can be used to copy or duplicate a Free Text string. To use, place the cursor over the Text and use COPY. The Text will be copied and the Menu will change to allow Rotation, Changing the Text Size and Horizontal or Vertical Justification. Complete the operation with F1 COMPLETE.

6.4 2-D LINES

COPY can be used to copy or duplicate a 2-D Line or a combined number of 2-D Lines. To use, place the cursor over the 2-D Line to be copied and select COPY. The 2-D Line will be copied and the Menu will change to allow Rotation and Mirroring.

6.5 HIERARCHICAL SYMBOLS

When a Hierarchical symbol is copied, the sheet that it references will also be copied. If the referenced sheet also contains Hierarchical symbols, then the sheets referenced by those symbols will also be opened, and so on, down the entire sub-tree of the hierarchy. Since this is such a drastic operation, the following prompt is issued:

COPY HIERARCHICAL COMPONENT
ARE YOU SURE (Y/N)?

Selecting "N" will quit the copy routine without copying the Hierarchical symbol. Selecting "Y" will allow the Hierarchical symbol to be copied, and when the copied symbol is placed, all the

sheets referenced either directly or indirectly, will be copied. If any numbered sheets are referenced, the copies will get new sheet numbers.

Chapter 7

THE MOVE AND DELETE ROUTINES

7.0 INTRODUCTION

The MOVE routine permits the moving and rotating, and sometimes X/Y mirroring, of any item selected with the cursor. The cursor selection is intelligent in that the system recognizes the item selected, and presents the correct Menu for the item selected.

The DELETE routine is used to Delete items from the Schematic. DELETE is also intelligent in that the system recognizes the item selected and presents the correct menu for the item selected.

7.1 MOVE

Specifically, the MOVE routine can "move":

- Parts/Gates
- Part Text
- Connections
- Tie Dots
- Hierarchical Symbols
- Text
- 2-D Lines
- Buses (without connections)
- Net Names
- Off-page References

With the exception of MOVE PART, MOVE CONNECTION and MOVE BUS, the MOVE routine is a very simple routine that moves the items selected. These three exceptions are described in more detail below.

7.1.1 MOVE PART

To use, place the cursor over the part and select F6 MOVE.

The Outline of moved part is attached to the cursor and the Menu Command changes to:

MOVE
1 COMPLETE
2 ROTATE
3 MIRROR X
4 MIRROR Y
5 ALTERNATE*
0 EXIT

*If Alternate Decal exists

The Menu Command performs exactly as the same menu command as described in ADD PART, Section 3.1 and will not be described at this point. Selection of F0 cancels the MOVE command. The connections to the Part will follow the part to its new location.

7.1.2 MOVE CONNECTION

7.1.2.1 MOVE SEGMENT

To use, place the cursor over the segment you wish to move and select F6 MOVE. Three operations are possible: MOVE SEGMENT, MOVE CORNER and DISCONNECT PIN. Be sure this segment is NOT tied to a pin, otherwise the DISCONNECT PIN feature will be invoked. The segment being moved turns WHITE and the menu command changes to:

MOVE
1 COMPLETE
2 CUT SEGMENT
0 EXIT

and the nearest Part Pin to which the connection is attached, together with the Net Signal Name, is displayed in the Yellow window.

Movement of the cursor results in:

1. Horizontal movement of a Middle Vertical segment,

2. Vertical movement of a Middle Horizontal segment,

F1 COMPLETE is used to finalize or complete the move.

F2 CUT SEGMENT is used to break a Vertical or Horizontal line segment into two parts and to manipulate either part at the break point.

To use CUT SEG, be in MOVE, place the cursor over the line to be cut and select F2 CUT SEG. The line will be picked and cut into two parts. The Menu window will be changed as shown below:

MOVE
CUT SEG
1 COMPLETE
2 SWAP CRN
0 EXIT

Moving the cursor perpendicular to the line will move one portion of the line segment that has been divided by the CUT SEG command. Movement of the cursor along the line moves the cut point of the line.

If you wish to move the second part of the line instead of the first part selected, use F2 SWAP CRN. This will cause the first line part to be moved to its original location and the Move capability will be transferred to the second part of the line.

F1 COMPLETE finalizes the location of the two line segments created by CUT SEG.

7.1.2.2 MOVE CORNER

To use, place the cursor over the corner you wish to move and select F6 MOVE. Be sure to select the corner exactly so you do not go into MOVE SEGMENT. The connections being moved will turn White and the menu window will be changed to as shown below:

MOVE
1 COMPLETE

2 ANGLE
0 EXIT

Moving the cursor will cause the corner as well as the two adjacent corners to more orthogonally. F1 COMPLETE is used to finalize the move.

F2 ANGLE is used if you wish to angle the adjacent segments. F2 ANGLE acts as a toggle switch, turning the ANGLE option On and Off.

7.1.2.3 DISCONNECT PIN

To use, place the cursor over the segment and select F6 MOVE. If the segment selected by the cursor is an End segment of the connection, i.e., one end terminates on a pin on a part or a tie dot, the connection is detached from the Part Pin or the Tie Dot and attached to the cursor and the connection will then follow movement of the cursor, and you are put in the ADD CONN routine to finish the connections (see Chapter 4, ADD CONNECTIONS for details).

7.1.3 MOVE BUS

Only the corner or segments of a bus can be moved and these can be done only if no connections are tied to the bus. If the whole bus is to be moved or the bus has connections tied to it, use the GROUP MOVE routine to perform the operation.

The MOVE SEGMENT and MOVE CORNER rotuines work the same as the equivalent operations for a Connection.

7.2 DELETE

DELETE recognizes the following items:

Parts/Gates
Connections
Buses (without connections)
On/Off Sheet Symbols
Text
2-D Lines

Net Names
Pwr/Gnd Symbols
Hierarchical Symbols

The DELETE routine will erase or delete various items from the schematic. DELETE is "Intelligent" in that you do not need to indicate the item to be deleted. Place the cursor over the item to be deleted, be in the top level routine, and select F7 DELETE. The system selects the item, recognizes it, temporarily erases from the CRT that which will be deleted and presents the user with the prompt:

CONFIRM DELETION Y/(N)?

An "N" will cancel the selection; a "Y" will complete the deletion.

7.2.1 NOTES ON USING THE DELETE ROUTINE

1. Deletion of a part will delete the part and disconnect the conductors connected to the part.

2. The deletion of an On/Off Sheet Tag also deletes the conductor associated with it.

3. Deleting the only Net Name associated with a Group of Connections causes the connections to have an auto net name $$$$ XXXX assigned to them.

6. Hierarchical Symbols: When a Hierarchical symbol is deleted, the user has the option to delete just the symbol itself or the symbol plus the sheet it references. If the user chooses to delete the referenced sheet, then any other sheets referenced by Hierarchical symbols on the referenced sheet will be deleted also, and so on, down through the entire hierarchical sub-tree. After the "Confirm Deletion" prompt, the following prompt will be issued to allow selection of the option to delete the referenced sheets:

DELETE ASSOCIATED SHEET (Y/N)?

If the associated or referenced sheet is not deleted, and it is a

Hierarchical sheet without a Sheet Number, it will be assigned the next unused Sheet Number and become a normal numbered sheet.

Chapter 8

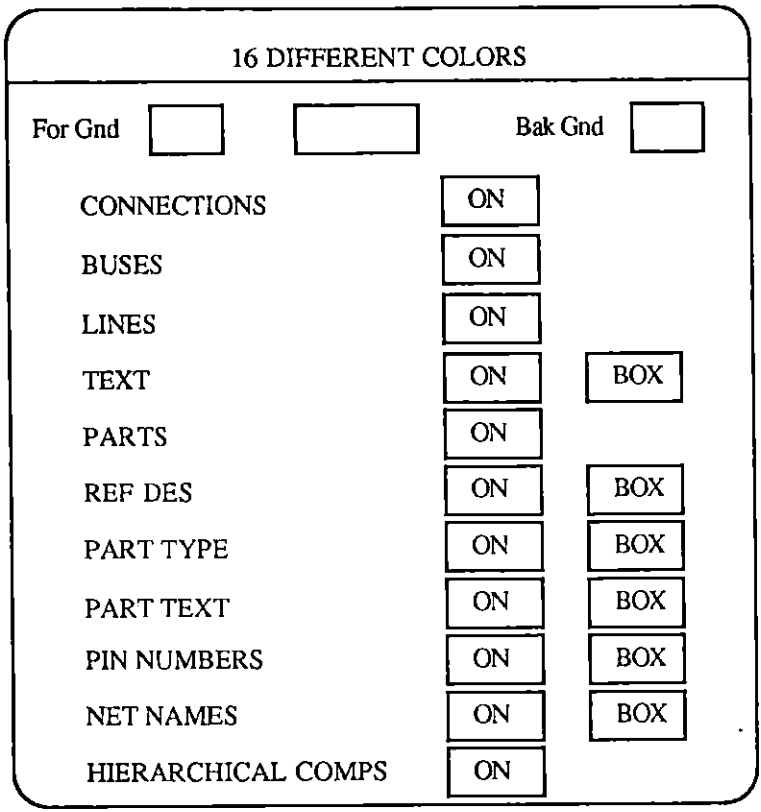
THE SETUP ROUTINE

8.1 INTRODUCTION

The SET UP routine is a multi-function routine in that a diverse number of functions can be activitated from within the routine. Specifically, the various functions include:

(1) setting the various colors for features (Ref Des, Decals, Connections, Parts, etc.) as displayed at the CRT
(2) setting "real" width of conductors/lines as displayed at the CRT
(3) setting the backup file interval
(4) setting the 2-D, conductor and bus line widths
(5) setting text size
(6) setting displayed dot grid spacing
(7) setting sheet size
(8) establishing setup pan criteria
(9) setting bus angle offset
(10) setting tie dot diameter
(11) using Macros

When F8 SETUP is selected, the CRT changes to the following:



CRT SETUP SCREEN

The Command Menu changes to the following:

SETUP
1 SELECT
2 PARAMETERS
3 MACROS
0 EXIT

8.2 CRT SETUP SELECT

The Menu Command F1 SELECT and Mouse Button 1 (left hand) are used to select the set up of the CRT screen. Refer to the previous figure (CRT Setup Screen) for the following discussion.

Along the top of the screen is a line containing 16 different colors (EGA). Any of the 11 items on the CRT, as well as the CRT Foreground and Background can be painted with any of the 16 colors.

To do this, place the cursor over the desired color box (top line) and using SELECT (left hand mouse key), select the color. When selected, the middle rectangle in the second line (unnamed) turns to the selected color, indicating the selection is complete. Move the mouse to the box associated with the item to be changed and click the left hand #1 mouse button. That box will be changed to the selected color and when the schematic is again displayed, the items in the schematic will be displayed in the selected color.

The second line on the CRT Setup Screen contains:

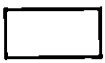
For Gnd Bak Gnd

The FOR GND Box pertains to the color you desire for the Foreground Text and other items used to view. The Setup Display changes its color in the same fashion as the schematic items.

The middle box (unnamed) is used to indicate the current color selected. The BAK GND Box permits the Background color of the CRT screen containing the schematic to be changed. Change the Background color in the same fashion as the schematic items.

The CRT contains 11 schematic items starting with CONNECTIONS and finishing with HIERARCHICAL COMPS. To change the color of any schematic item, touch the ON/OFF Box rectangle associated with the item, with the mouse cursor, after the menu color has been selected, and check that the Box changes to the selected color.

Six of the schematic items (Text, Ref Des, Part Type, Part Text, Pin Number and Net Name) have the word "Box" associated with them. The word "Box" is a switch that permits a Box



to be drawn around the item when displayed at the CRT. This Box serves two purposes: (1) Indicates the exact size of the item when plotted, thereby helping while moving or placing the item, and (2) Provides better visibility and identification of the item. To turn the Box ON, select one of the 16 EGA colors, move the cursor over the word "Box" and click the mouse left hand button.

To turn OFF a schematic item or to switch off the Box around an item, the procedure is to make it invisible by using the same color on the item as is being used for the Background color. When one of the 8 Schematic items is turned off (not displayed), its ON/OFF rectangle is made invisible and the word inside the rectangle is changed to OFF. When the Box associated with Text, REF DES, Part Type, Pin Number and Net Names is turned OFF, the Word "Box" remains, however, the Box figure around the word "Box" is made invisible and does not show up on the Schematic.

NOTE 1: We suggest you set up a combination of schematic item colors as well as FOR GND/BAK GND colors and continue to use your selections on all schematics. When the Setup is made for a schematic, it remains with that schematic for its entire life cycle, unless changed.

NOTE 2: The Setup selection for displaying items or making them invisible does not affect plotting of the schematic in any way.

8.3 PARAMETERS

F2 PARAMETERS is used to set up a wide variety of parameters associated wtih a schematic.

These Parameters are initially set to a Default value in the software, however, you may change them for a specific schematic. The new parameters stay with that schematic during its lifetime unless changed.

To use, be in SETUP and select F2 PARAMETERS. The CRT presentation will change to as follows:

DOT GRID	REAL WIDTH
BACKUP INTERVAL	SHEET SIZE
PAN ENABLE	PAN SENSITIVITY
PAN DELAY	PAN DISTANCE
BUS LINE WIDTH	BUS ANGLE OFFSET
CONNECTION WIDTH	TIE DOT DIAMETER
NET NAMES HEIGHT	LINE WIDTH
PIN NUMBERS HEIGHT	LINE WIDTH
PIN NAMES HEIGHT	LINE WIDTH
REF-DES HEIGHT	LINE WIDTH
PART TYPE HEIGHT	LINE WIDTH
OTHER TEXT HEIGHT	LINE WIDTH
DECAL	LINE WIDTH
2-D ITEMS	LINE WIDTH

(Associated with each item, are the legal limits for the item.).

To change an item, select the item by placing the cursor over its Value and clicking the Left mouse button. Use the Keyboard and Backspace to edit the VALUE. F0 EXIT completes the action.

8.3.1 **DOT GRID**

The DOT GRID subroutine of the SETUP routine is used to change the dot spacings between the Dot Grid displayed at the CRT. Note that the Dot Grid is a visible guide for drawing lines, decals, placing pads, etc. It is NOT related to the System Grid (sometimes called the "Snap Grid").

The default value for the spacing between the CRT dot grid is one inch or 1000 each 0.001" units, however, this can be changed by the designer using DOT GRID. To change the DOT GRID, select the DOT GRID entry. Type in the new value in units of 0.001". For example, if you wish spacing to be 2.0", type in 2000<CR>. The CRT will be repainted with the dots spaced at two-inch intervals.

If you should find yourself without a grid present, it is undoubtedly a result of contracting the scale of the CRT so that more dots are required to fill the area than there are dots available. Change the scale to decrease the area, and Dots should appear.

8.3.2 REAL WIDTH

The purpose of REAL WIDTH is to allow Lines to be displayed at their "real width" at the CRT, i.e., 12 mils wide, 50 mils wide, etc.

The option functions as follows: In response to REAL WIDTH, enter the width of a Line, above which all Lines using this width or wider will be displayed at the current real width. Lines with a width less than the selected width will not have True Width displayed, instead, the CRT displays only the center of the Line.

Example: Your Schematic contains some Lines at 10 mils wide and some Lines at 20 mils. You wish to see 20 mil Lines at their true width, but not 10 mil Lines. Select REAL WIDTH and type in 20 <CR>. The display will show Lines with Width below 20 mils with their center line only, and any lines at 20 mils or greater at their Real Width.

The rationale behind allowing the Engineer to select the widths to be displayed at their real width is to permit the user to "trade-off" between CRT paint time vs. content of the displayed information. Repaint of the CRT display, using Line center line display only, is somewhat faster than when using REAL WIDTH.

NOTE: The REAL WIDTH selection, in addition to controlling the width of the Line displayed at its real width, also controls the width of Text, 2-D lines, Decal Outlines, etc.

8.3.3 BACK UP INTERVAL

The BACK UP Routine (subroutine: SETUP, PARAMETERS) automatically acts to make a Backup file of the database in memory at intervals of time selected by the Engineer. When this occurs, the CPU message center window displays the word: "WORKING" during the few seconds the backup file is being placed on hard disk. When the backup file is made, it overwrites the previously made

backup file on disk.

The purpose of the Backup file is to give you a convenient way of recovering your work investment, in the event of an emergency such as a power failure, hardware failure, or corruption of your Job in the Job Files. The Backup file is a Binary .JOB file maintained in the LOGFIL subdirectory under the name CAE.SCH.

NOTE: This backup file is not to be confused with the backup file created by a DOS Editor or any other editors you may have on your system. It can be identified by its Name (CAE.SCH).

When BACK UP is selected, the message:

BACKUP JOB TIME INTERVAL (MINUTES) [5]>

appears. This message is a two-part message. First, it is telling you that the default interval for automatically making a backup job file. To change this interval to any time between 1 minute and 30 minutes, select BACKUP Interval and type in the minutes for the new desired interval and strike the Carriage Return.

Note that our experience has shown that taking a backup file every 5 minutes is an optimum time for minimizing the loss of data. We, therefore, recommend that you leave the job interval time set at 5 minutes.

Another way of making or "forcing" a Backup file is to select Job Out of the IN/OUT routine, and then strike the Carriage Return. This will create a Backup file, overwriting the previous Backup file. To recall the Backup file into memory, simply use the IN/OUT routine, select JOB IN (key 1) and strike the <CR> in response to the prompt for the Job Name.

8.3.4 SHEET SIZE

Sheet sizes of A through E are available as follows:

A size	8.5" x 11"
B size	11" x 17"
C size	17" x 22"

D size	22" x 34"
E size	34" x 44"

Enter the size desired. Additionally, for International use, Sizes A0-A4 are also available.

8.3.5 PAN

8.3.5.1 INTRODUCTION

AUTO PAN is a CRT manipulation function that is very useful. Specifically, with AUTO PAN, the Engineer can touch the right, left, top, bottom edge of the CRT and the display will automatically move away from the touched edge, as long as the edge is being touched and the data base size limits have not been exceeded.

8.3.5.2 PAN ENABLE Y/N

PAN ENABLE Y/N is a toggle that toggles the Automatic Pan On and Off. Each time PAN ENABLE is selected, the toggle position is changed and the PAN ENABLE line indicates the status.

8.3.5.3 PAN SENSITIVITY

This parameter setting determines how near the edge of the CRT the cursor must be before PAN is activated. A high setting (10) requires the cursor to be at the extreme edge of the CRT detection area, a low setting of 1 will activate PAN somewhat removed from the extreme edge. A setting of 5 is the Default.

8.3.5.4 PAN DELAY

PAN DELAY is used to adjust the time interval between the time the edge of the CRT is touched with the cursor and the "PAN" is executed. The Numbers are approximately 0.01 second increments. A setting of 10 represents 0.1 seconds; a setting of 1 represents 0.01 seconds.

We suggest you may not want to use extremely low settings such as 1, since each time you inadvertently touch the edge of the CRT with the mouse, the CRT will "instantly" pan. Our experience suggests

that the Default setting (.05 seconds) is probably the most practical selection.

8.3.5.5 PAN DISTANCE

PAN DISTANCE is used to set up the distance at the CRT of a single pan move action. The Numbers in ( ) are percentages of the Width and Height of the CRT that will be made by a single pan action. A selection of 50 will result in a pan moving the CRT display 50% of the width or height of the CRT. The Default is 10%. You should experiment to determine that Pan Distance that most nearly fits your mode of operation.

8.3.6 BUS LINE WIDTH

BUS LINE WIDTH is used to change the original width or thickness of lines used when creating a Bus. The Default is 25 mils. You may, by using BUS LINE WIDTH, change Width from 1 mil to 250 mils. The Bus is shown at its True Width if REAL WIDTH of PARAMETERS is set accordingly.

8.3.7 BUS ANGLE OFFSET

BUS ANGLE OFFSET is used to determine the point (in mils) at which a connection being connected to a bus, starts the Angle used to connect to the Bus.

8.3.8 CONNECTION WIDTH

CONNECTION WIDTH is used to set the Width (in mils) of connections on the Schematic.

8.3.9 TIE DOT DIAMETER

A Tie Dot is Automatically placed where a connection is tied to another connection. The Diameter of this Tie Dot can be changed by changing the Value in mils of the Number associated with the TIE DOT DIAMETER.

8.3.10 NET NAMES - HEIGHT (mils) LINE WIDTH (mils)

Each Net has a Net Name. The Height of the letters used to show the Net Name is established by HEIGHT (in mils). The Width of the Line used in the Net Name letters is established by NET NAME LINE WIDTH.

8.3.11 PIN NUMBER

PIN NUMBER is used to establish the Size (Height) and Line Width of the Number associated with the terminals on Part Decals. Terminals or Pads are those features contained within a Part Decal to which connections or conductors are attached. Height is the height of the Pin Number (in mils); Line Width is the width of the Line used in drawing the Pin Number.

8.3.12 PIN NAMES

PIN NAMES is used to establish the Size (Height) and Line Width of the Pin Name Text used on a Part Pin to provide a Pin Name (NOT a Pin Number).

8.3.13 2-D ITEMS

2-D ITEMS is used to set the Default Line Width of 2-D Lines. When Lines of different Line Width are created on the schematic, the entry of 2-D ITEMS is expanded to allow for additional Line Width.

8.4 MACROS

The creation and usage of Macros is the subject of a complete chapter devoted to Macros. See Chapter 13, The MACRO Routine.
16

Chapter 9

ADD BUS ROUTINE

9.0 INTRODUCTION

The ADD BUS subroutine is located in the OTHER MENU Routine.

The major routine of OTHER MENU is an assortment of other menu commands that have been grouped together under OTHER-MENU. When OTHER MENU is selected, the menu changes to the following:

OTHER MENU
1 ADD BUS (Chapter 9)
2 ADD TEXT (Chapter 10)
3 2-D LINES (Chapter 11)
4 GROUP (Chapter 12)
5 COPY (Chapter 11)
6 MOVE (Chapter 7)
7 DELETE (Chapter 7)
8 SETUP (Chapter 8)
9 HIER/LIB (Chapters 13, 14, 15)
0 EXIT

9.1 ADD BUS

A "Bus" is a collection of conductors that are grouped together into a single graphical symbol (a special thickened conductor). Usually the conductors that have been graphically "shorthanded" into a single Bus symbol are closely related, such as an address line. Putting in a Bus is similar to putting in a conductor in that they can be added, named, modified, connected to and deleted.

A Bus must have a Name associated with the bus before its entry is complete, using the format:

BUS AB [N_1:N_2]

where AB is the bus stem name, N_1 is the lowest bit and N_2 is the highest bit.

To add a Bus, you must be in the OTHER MENU routine and select F1 ADD BUS. The menu changes to:

OTHER MENU
ADD BUS
1 ADD CRN
2 DEL CRN
9 COMPLETE
0 EXIT

A White line starts at the location of the cursor at the time F1 ADD BUS was selected. Movement of the mouse/cursor drags the Bus Line Up, Down, Right, Left from the start point.

F1 ADD CRN allows the change of direction of the Bus, in either an "X" or "Y" direction. Note that the corner will be composed of 2 corners set at a 45 degree angle.

F2 DEL CRN deletes the last corner put into the Bus. DEL CRN is successive, i.e., all corners in the entire Bus may be removed by a successive DEL CRN selection.

F9 COMPLETE is used to terminate the drawing of the Bus symbol and fix it in position. Upon selection of F9 COMPLETE, the Bus will be displayed at the CRT in the conductor color and as a thickened line. You must respond to the prompt:

BUS NAME>

The Bus Name format consists of two parts: (1) the Name of the Bus (alpha characters), and (2) the Bit Designation that will be contained within the Bus **[N_1:N_2]** where N_1 is the lowest Bit Number and N_2 is the highest Bit Number. The total Bus Name, including Bit Number, may not exceed 12 characters. (The Bit Number range is from 0-99.) An error message will result, and the name will not be accepted if the format is not followed.

When the Name is accepted, a White Name Box is displayed. F2 MOVE allows the Bus Name to be moved or rotated, either as an Up-Down or Left-Right reading position.

F1 COMPLETE completes the entry of the Named Bus into the schematic.

9.1.1 NOTES ON BUSES

1. Note that a Bus cannot be ended on a Connection, Terminal or Unnamed Bus, but is ended in space.

2. A Bus must have a Name before a Connection can be connected to it.

3. A Bus Name can be changed by using the EDIT TEXT routine.

4. A Bus that appears on two or more sheets using the same Bus Name will be tied together when the Net List is generated.

5. The Width of a Bus (in mils) can be changed using SETUP, PARAMETERS and setting the Bus Line Width to the desired Width. To display the Bus at its True or Real Width, use SETUP, PARAMETERS and set the Real Width to the Bus Width or a number smaller than the Bus Width.

6. The Angle Length used in Buses can be modified in SETUP, PARAMETERS.

7. The Routine MOVE can be used to move corners or segments of a Bus pattern IF No connections have been made to the Bus.

8. If connections have been made to the bus or you wish to move the whole bus, use GROUP and create a group that contains the Bus and Connections that are to be moved. After the "Group" has been created, use GROUP MOVE to relocate the bus and its connections.

Chapter 10

ADD TEXT ROUTINE

The routine of F2 ADD TEXT located in OTHER MENU is used to add lines of free text on to the schematic. Free text is that text such as Notes, Comments that are not associated with other items, such as in the REF DES for a Part, or the Signal Name for a Connection.

To use, be in the OTHER MENU routine and select F2 ADD TEXT and respond to the prompt:

NAME>

with the Text string to be entered. At the Carriage Return, a White "Text Box" containing the undisplayed text will be attached to the cursor. The size of the Box is the actual size of the free text when plotted. The Text Box may be moved around on the schematic by movement of the mouse.

F2 ROTATE will rotate either Left-to-Right or Up-to-Down reading.

F3 SIZE permits the changing of the Text Size. When selected, respond to the prompt:

CHANGE TEXT SIZE [HHH WW]>

with the new desired size. Use the format of HHH WW, where HHH is the Height in mils and WW is the Line Width used to plot the text. The Carriage Return completing the size change results in the Text Box being changed to the new size.

F4 HORIZ JUST permits the Right, Left or Middle justification of the Text (in the X direction).

F5 VERT JUST permits the Up, Down and Middle justification of the Text (in the Y direction).

F1 COMPLETE completes the entry of the Free Text into the schematic.

Chapter 11

2-D LINES ROUTINE

11.0 INTRODUCTION

The routine of 2-D LINES located in the OTHER MENU routine permits the adding, modifying and deleting of, as well as the calling up or storing 2-D Lines from the Library. 2-D Lines are used as a Drafting tool when non-conductive lines are required on the schematic.

When F3 2-D LINES is selected, the menu changes to the following:

OTHER MENU
2-D LINES
1 NEW POLY
2 NEW CIRCLE
3 MODIFY
4 COMBINE
5 COPY
6 MOVE
7 DELETE
8 FROM LIB
9 TO LIB
0 EXIT

When making 2-D Lines, remember to check the Default value for Grid and Width. You probably will want to use a Grid resolution of 10 and a Width appropriate for the Draw Line being created.

NOTE: The 2-D Lines are displayed at their true width if the Real Width of SETUP, PARAMETERS is set equal to or less than the 2-D Line Width.

11.1 NEW POLY

To start a new polygon, be in the 2-D LINE option, place the cursor exactly where you wish the drawing line to start and select F1 NEW

POLY. The Menu window changes to that as shown below:

OTHER MENU
2-D LINES
NEW POLY
1 ADD CRN
2 DEL CRN
3 ANGLE
8 END
9 COMPLETE
0 EXIT

To use, place the cursor exactly where you wish the 2-D Line to start and select F1 NEW POLY.

A White line is drawn starting from the cursor position and is displayed at the CRT. Use the options of ADD CORNER, DELETE CORNER, ANGLE to create the drawing line, and if it is to be a closed polygon, when in the vicinity of the start point, select COMPLETE for Auto Complete. If the drawing line is to be ended prior to the Start point, use END to terminate the line at the current location of the cursor.

11.2 NEW CIRCLE

To create a circle rather than a line or polygon, be in the 2-D LINE subroutine and select F2 NEW CIRCLE, after placing the cursor on the circumference of the circle to be created (the start point). Move the cursor away from the start point. The cursor movement distance and direction determines the circle diameter and its direction from the start point.

ANGLE permits the direction of the circle diameter to be at any angle down to 1° from the start point.

LINE WIDTH permits changing of the circle Line Width.

When the circle is at the correct diameter and at the correct location, enter the circle into the database by selecting COMPLETE.

11.3 **MODIFY**

The F3 MODIFY command, in 2-D LINES, is used to modify an existing 2-D Line. To use, be in 2-D LINES, have a 2-D Line on the screen. Place the cursor over the 2-D Line to be modified and select F3 MODIFY. The 2-D Line will be selected (turns White) and the menu changes to the following:

OTHER MENU
2-D LINES
MODIFY
1 COMPLETE
2 MOVE ARC
3 CUT SEG
4 LINE WIDTH
6 LINE STYLE
0 EXIT

F2 MOVE ARC moves the selected Line, creating an Arc as the mouse is moved. Horizontal Lines move in the Vertical direction; Vertical Lines move in the Horizontal direction. Diagonal Lines move in all directions unless anchored by a Line at the End of the diagonal. MODIFY, NEW CIRCLE changes the Diameter of the circle and its direction of creation. (MODIFY, NEW CIRCLE has a slightly different menu structure from MODIFY, NEW POLY.)

F6 LINE STYLE permits changing of a 2-D Line or polygon (Not Circle, Arc) from a Solid line to a Dotted line and back.

11.4 **COMBINE**

F4 COMBINE permits the combining, into a single entry, more than one 2-D Line item as well as Free Text items. When combined, these items are treated as a single 2-D Line item, and all combined items can be moved together, deleted together, etc.

To use, select F4 COMBINE. The menu changes to allow ADDING items to the combined items, or DELETING if the intention is to de-combine. Select F1 ADD ITEMS or F2 DEL ITEMS as appropriate. Thereafter, place the cursor over the Line or Text and select F1 SELECT. The selected Line will change color to

White, while Text will temporarily disappear. Continue to use F1 SELECT to select the items to Add or Delete from the combined group.

NOTE: When combining 2-D Lines and Text, always select the 2-D Line first, i.e., combine Text with previously selected 2-D Lines.

NOTE: COPY, MOVE and DELETE are the same functions as contained at a higher level in the Menu structure, and are included in the 2-D LINE Routine for user convenience.

11.5 COPY

The F5 COPY command in 2-D LINES is used for copying or duplicating 2-D Lines, Parts and Text (see Chapter 6)

11.6 MOVE

F6 MOVE is a multi-application command and allows any item in the schematic to be moved (see Chapter 7).

11.7 DELETE

F7 DELETE is used to delete any item contained on the schematic (see section 7.2 of Chapter 7).

11.8 FROM LIB

F8 FROM LIB is used to bring a 2-D Line item in from the Library, that has been previously created and saved, and add the item to the current schematic.

To use, select F8 FROM LIB and respond to the prompt:

ADD ITEM WITH NAME>

by entering the name of the previously created 2-D item <CR>

The word "Working" will appear in the Yellow window, after which the retrieved item will be attached (in White color) to the cursor and the menu will change to the following:

OTHER MENU
2-D LINE
FROM LIB
1 COMPLETE
2 ROTATE
0 EXIT

Place the 2-D item where desired, rotating it if you wish, and place it with F1 COMPLETE.

11.9 TO LIB

F9 TO LIBRARY is used to save a 2-D item to the Library for subsequent use on other sheets of the same schematic, or an entirely different schematic.

To use, place the cursor over the 2-D item to be saved to the Library (combine a number of 2-D items and Text if required) and select F9 TO LIBRARY and respond to the prompt:

SAVE ITEM WITH NAME>

with the Name of the 2-D item to be stored in the Library <CR>. As the item is saved, the Yellow window will show "Working."

Chapter 12

GROUP ROUTINE

12.0 INTRODUCTION

The GROUP facility contained in the OTHER MENU Routine permits the designation of an area containing Parts and Connections, and moving this group to an alternate location on the same sheet, or Saved and placed on another sheet in the same Design, or used in a different design. GROUP may be used in the construction of a Hierarchical Design. Finally, GROUP is the rotuine used to move a Bus that has connections made to the Bus.

Items can be added or deleted from the group. In addition, a Group can be Cut to the hard disk and Pasted into another Design.

To use, be in the OTHER MENU routine and select F4 GROUP. The menu will change to:

OTHER MENU
GROUP
1 DEFINE GRP
2 PASTE GRP
0 EXIT

12.1 DEFINE GROUP

To define the group, place the cursor at the lower left corner of the items to be grouped and select F1 DEFINE GRP. The F1 DEFINE GRP will be replaced with F1 COMPLETE. Move the cursor to the upper right corner and select F1 COMPLETE to enclose the group area. The group area will be highlighted and the menu will change to:

OTHER GROUP
GROUP
DEFINE GRP
1 MOVE GRP

2 COPY GRP
3 DELETE GRP
4 CUT GRP
8 ADD ITEM
9 DELETE ITEM
0 EXIT

F1 MOVE is used to move the group to the position specified by the cursor.

NOTE: Conductors inside a group area will retain their position after GROUP MOVE. Conductors going outside the group will be stretched to the new group location.

F2 COPY copies the contents of the Group and places the copy at the cursor location. Component REF DES names are automatically incremented, such that duplicate REF DES will not exist after the "copy" is made. Spaces or blanks in the REF DES sequence are filled.

This COPY GROUP feature is useful for "Step and Repeat" within the limits of available memory in your system.

The prompt:

CUT CONNECTIONS AT BOUNDARY (Y/N)?

is handled as described in NOTES below.

F3 DELETE, when selected, will delete the contents of the Group after responding to the confirm prompt.

F4 CUT is used to save the contents of the Group to hard disk as a file. This file can be recalled and used in the the same schematic or an entirely different schematic.

When F4 CUT is selected, the following prompt is given:

ENTER GROUP FILE NAME>

Respond with any name. Respond to the prompt:

CUT CONNECTIONS AT BOUNDRY (Y/N)?

The group will be saved with the name and extension .GRP, in the \PADS\LOGFIL directory.

F8 ADD ITEM permits the adding of an item (a Part and its connections, Free Text, 2-D Line, an Off-Page reference) that is located outside the group area, to the group. Connections can only be added to the group if there is at least one corner within the group box. Buses can only be added if part of a bus segment lies within the group box. Place the cursor on the item and select F8 ADD ITEM. The item will be highlighted indicating that it has been added to the group. When the group is moved, the item that has been added will retain its position relative to the group, after GROUP MOVE has been completed.

F9 DELETE ITEM permits the deleting of any item inside the group from the group. After the group has been created, place the cursor on an item within the group and select F9. The item to be deleted will change color indicating it has been deleted from the group. If the group is now moved, the deleted item will not move with the group. Connections tied to items within the group cannot be deleted.

NOTES FOR GROUP CONNECTIONS AND PARTS:

The following rules apply to Parts and Connections that have been cut in GROUP COPY, CUT and DELETE

1. If a connection pattern is such that it leaves AND returns to the Group, the connection will NOT be cut; instead, the entire connection will be included in the New Group.

2. A connection that is entirely within the Group being created, and thus is NOT cut at the Group Boundary, will be included in the New Group. Unnamed sub-nets will be assigned a new system generated net name of the form $$$NNNN when the group is copied or cut and pasted.

3. If a connection is cut by the Group Boundary and the response to

the prompt "CUT AT BOUNDARY" is Y:

(1) The part of the connection within the New Group will be included in the New Group.

(2) An Off-Page Symbol will be automatically placed at the Boundary Cut location.

(3) If the connection is unnamed, a unique net name of the form ###NNNN will be attached to the Off-Page symbol.

(4) You must tie these connections into the Design by editing the Net Name, and Renaming as appropriate.

(5) If the Name of the Cut Connection is a Designer assigned name or Bus Bit Name, the Same Name will be reused in the New Group.

(6) If Deleting a Group, that part of the Cut Connection within the Design but outside the Boundary, will be retained with an automatically assigned Off-Page Symbol. Unnamed connections will be named uniquely with a net name of the form ###NNNN.

4. If a connection is cut by the Group Boundary and the response to the prompt "CUT AT BOUNDARY" is N:

(1) The part of the connection falling within the Group will NOT be included in the New Group.

(2) If Deleting a Group, the part of the connection in the Design but outside the Group will also be deleted.

5. The connections that are contained in a Group that has been created from another Group, i.e., a second generation group, and have been "cut" at the Group Boundary, will use the SAME Name as the Group from which it was created. The result of this, of course, is to tie these connections together in the Net List. The Designer must change the Net Names, if this combining is NOT desired.

6. If the Group Outline includes a portion of a Part, its entire part (or gate) will be included in the New Group.

7. When a Part is included in a Group, including Second Generation Groups, the REF DES is always upward incremented. Any blank or unused REF DES Names are used in the automatic REF DES Assignment.

12.2 PASTE

When in the subroutine GROUP OPER, the selection of F2 PASTE permits the "pasting" a named group file that had been previously saved as a "CUT" in memory.

To use, select F2 PASTE and respond to the prompt:

ENTER GROUP FILENAME (CR=CAE.GRP)>

with the name used to CUT and SAVE the group to disk. The CUT is recalled from the \PADS\LOGFIL directory and placed at the cursor location.

The Rules for handling Connections and Parts for GROUP PASTE are the same as for GROUP COPY (see NOTES FOR GROUP CONNECTIONS AND PARTS above).

12.3 GROUPING OF HIERARCHICAL SYMBOLS

The GROUP routine can also include a Hierarchical Symbol. When a Group is created that includes a Hierarchical symbol, a Warning message will be issued as follows:

WARNING: GROUP CONTAINS HIERARCHICAL SYMBOLS

You should be aware that when you include a Hierarchical symbol in a Group, the circuitry represented by this Hierarchical symbol will also be included in the Group and when pasted into another design, will include the circuitry associated with the symbol.

12.4 GROUP DELETE WITH HIERARCHICAL SYMBOLS

When a Group is being created for deletion of the Group, and this group includes a Hierarchical Symbol, the normal "Group Delete" message prompts are issued, along with a special prompt which follows:

DELETE ASSOCIATED SHEETS (Y/N?)

A "Y" response will result in the sheets represented by the Hierarchical symbol being deleted from the design.

An "N" response will delete the symbols but not the circuitry.

Chapter 13

THE PARTS LIBRARY MANAGER - PLIB

13.0 INTRODUCTION

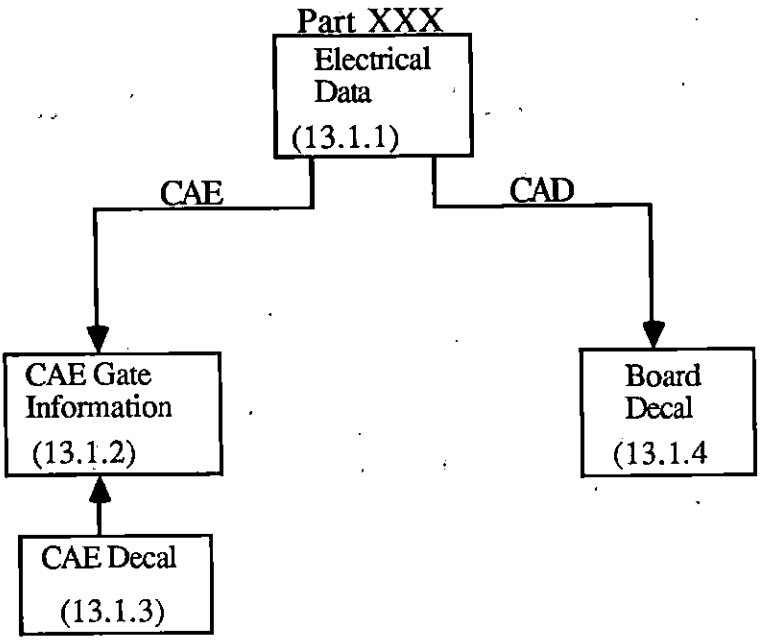
In Part III, the ***PADS-LOGIC*** Reference Manual, there are two (2) chapters devoted to the ***PADS-LOGIC*** Parts Library. The first is this chapter (Chapter 13), which describes the Parts Library Manager (PLIB) together with the organization of the ***PADS-LOGIC*** and ***PADS-PCB*** Sublibraries, as well as the Users Library and the Standard Library. The second chapter is the following chapter (Chapter 11) that provides complete details on the creation of an individual part.

13.1 THE ANATOMY OF A *PADS-LOGIC* PART

A Part as contained in ***PADS-LOGIC*** is an Integrated CAE/CAD Part. That is to say, the Part contains all the information needed for CAE (***PADS-LOGIC***) as well as for Board CAD (***PADS-PCB***). If your Board CAD system is not ***PADS-PCB***, then you use the CAE portion of the ***PADS-LOGIC*** Part, and the resulting ***PADS-LOGIC*** Parts and Net List will call out the Part as appropriate for your Board CAD system. Naturally, you will need to insure the ***PADS-LOGIC*** Net List correctly identifies the Part as necessary for your Board CAD system. This can be done in two ways:

1. Editing the ***PADS-LOGIC*** Net List for your Design, or
2. Modifying the ***PADS-LOGIC*** Library parts to call out your Board CAD Part/Decal rather than the ***PADS-PCB*** Part/Decal.

The structure of a ***PADS-LOGIC*** Part is as shown in the following Block Diagram:



13.1.1 ELECTRICAL DATA

The Electrical Data for a Part contains the Electrical Data as used by BOTH the CAE and CAD system. In the process of creating a new part, the user compiles Electrical data using Vendor specifications as the primary source data. Contained in the Electrical Data are the following items:

1. PART NAME (A Prefix and Suffix)
By using a combination of Prefix and Suffix, a Single Part Entry can be used for all the IC's of a particular family. For example, a single part entry can be used to describe the 04 family of Inverters: 5404, 54ALS04, 54H04, 54L04, 54LS04, 54S04, 7404, 74ALS04, 74H04, 742S04 and 74S04.

2. THE PCB DECAL NAME
This is a "Pointer" that appears in the Net List that is sent from ***PADS-LOGIC*** to the Board CAD System and is used to call up the correct Decal or footprint when designing the PCB Board.

3. LOGIC FAMILY
Used in the Automatic REF DES Assignment (PKGTYP.DAT) of the schematic.

4. ATTRIBUTES (Optional)
There are up to ten Attributes that may be assigned to a Part. The user may assign any type of attribute he wishes to a part. Most of the parts in the Library have been assigned five attributes. These are:

(1) Description: Example: 2-Input Positive NOR
(2) Cost: $
(3) Part Name: (Also can be used for Company Part Number)
(4) Manufacturer #1
(5) Manufacturer #2

Discrete parts will have a "Value" attribute, i.e., 10K Ohms and may also have a "Tolerance" attribute, i.e., ±5%, etc.

The attribute line must conform to the following format: an attribute name of 16 characters or less, i.e., "Value," "Description," etc., followed by a colon ":", then followed by an attribute value which is any alphanumeric string.

5. DISPLAYED ATTRIBUTE
Any two attributes may be displayed at the CRT and plotted.

6. NON-NUMERIC PIN
Provides a conversion for those Parts that use Alphanumeric rather than Numeric Pins, i.e., A1, A2, etc.

7. SIGNAL PINS
Permits the "building in" to the Part the Pin Number, and Track Width of Standard Power and Ground Data. When the Part is used in the Schematic, a Power and Ground Net is automatically generated.

8. NUMBER OF GATES
Indicates the number of Gates in the Part.

NOTE: "Gates" is a "generic" term and is NOT confined solely to

Gates in a multi-gate IC. While a 74LS04 has six gates, a 68000 has one "gate," a Discrete Part (a Resistor) will have one "gate," etc.

9. TYPE OF GATE
Used to indicate the "Swappability" of a gate with other similar gates in the same component or other identical components on the board, when commanded to Swap by an ECO, or Back Annotation File from the Board CAD System.

10. GATE DECAL
Used to identify by Name the CAE Decal that will be used by the Part on the schematic.

11. ALTERNATE GATE DECALS
Used to identify the DeMorgan Equivalent or Alternate Part Decal.

13.1.2 GATE INFORMATION

The "Gate" information for a Part is contained in a graphical CRT presentation created during the creation of a Part. The CRT presentation contains the following:

1. There is a Decal for each Gate and DeMorgan Equivalent Decal in the Part. A 74LS1000 Gate CRT screen would contain 8 Gate Decals (4 for the Primary and 4 for the DeMorgan Equivalent) in the CRT presentation. An IC without "gates" and represented at the Component level (Example: 68000) would have a single gate. A Resistor or Capacitor would have a single gate with alternate representations.

2. Correct Pin Numbers assigned to each gate.

3. The correct Gate Modifier, i.e., U1A, U1B, etc.

4. The Pin Type for each Pin, i.e., Source, Load, Bidirectional, etc.

5. The Swappability of the Pins within the Gate.

13.1.3 SCHEMATIC (CAE) DECAL

The CAE Decal consists of the Graphical information contained in the Gate Decal. A large number of precreated Decals are supplied with ***PADS-LOGIC***. In addition, an Editor is provided for use in creating new Decals. A CAE Decal consists of the following:

1. The Graphical Symbol displayed on the Schematic.
2. Reserved Text Boxes for:
   (1) Pin Numbers
   (2) REF DES
   (3) Pin Type
   (4) Pin Swappability
   (5) Part Type (where used)
   (6) An Attribute
   (7) An Attribute

13.1.4 PCB DECAL

The PCB Decal is the footprint or Decal used for the Part when in the Board CAD System. In ***PADS-PCB***, the PCB Decal consists of:

1. The Footprint including Terminal Pads and Component Outline
2. Pad Numbers
3. Decal Name

13.2 PLIB

This Section discusses the contents and use of the System Parts Library Manager and its Editor (PLIB).

Access to the System Parts Library Manager is through the DOS PADS DIRECTORY Path or the PADS Shell.

To access the Library Editor from the PADS Graphical Shell, use the mouse cursor and select the ***PADS-LOGIC*** PLIB port.

To access the Library Editor from the DOS Directory Path, in response to C:\PADS>

Respond: PLIB <CR>

The Editor associated with the System Parts Library will appear, as shown in the following Figure 13.1:

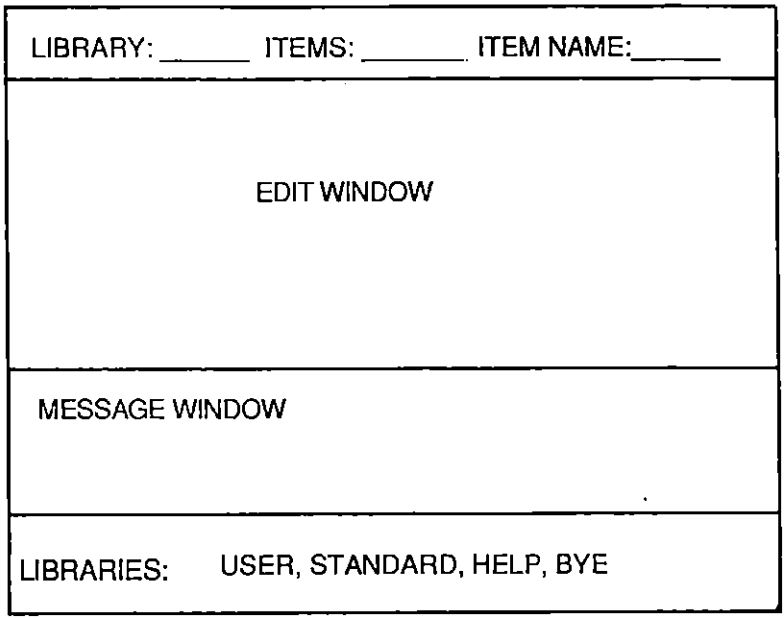


Figure 13.1 - LIBRARIES

There are two Library areas associated with ***PADS-LOGIC***: 1) The USER area and 2) the STANDARD area. The User Library area contains the part decals, part types, 2-D lines, and CAE Decals created by the designer used and stored in the User area. It is selected by typing U.

The Standard Library area contains the 2-D lines, part decals, CAE Decals, and part types shipped with the ***PADS-LOGIC*** Software. It is selected by typing S. After selecting either the User area or the Standard area, the CRT presentation is as shown in the following Figure 13.2:

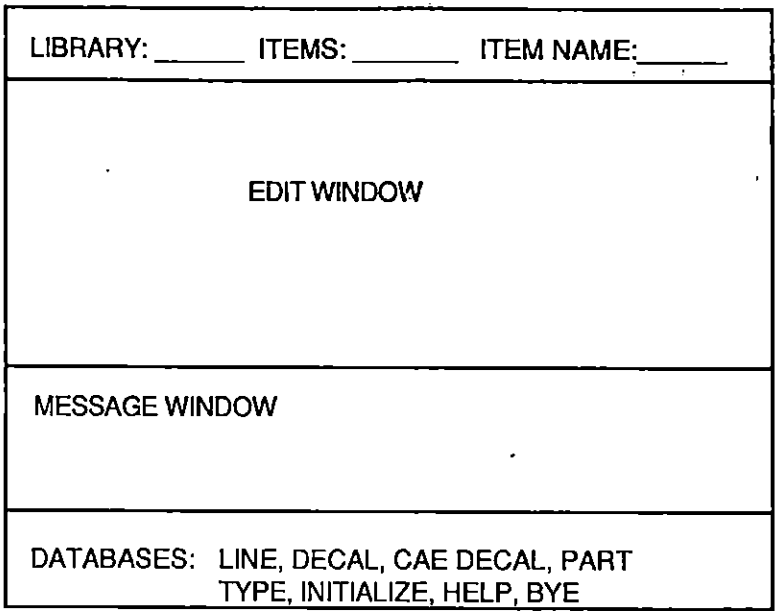


Figure 13.2 -PLIB EDITOR

Four databases are contained in both the System User Library and the System Standard Library. These are: (1) 2-D Lines, (2) PCB Decals, (3) CAE Decals, and (4) Part Types.

13.3 2-D LINES DATABASE

Lines are items constructed using the subroutine 2-D LINES. Typically, Schematic Sheet and PCB Board Outlines are composed of lines only and stored in the LINES Database of the Standard Parts Library.

13.4.1 PCB DECALS DATABASE

A PCB Decal is the PCB footprint of a component as used to display the component at the CRT or to plot the component in ***PADS-PCB***. All PCB Decals created by the user in the ***PADS-PCB*** program are saved to the PCB DECAL database of the User area in the System Parts Library. A PCB Decal can be "forced" into either the User area or the Standard area at the time of its creation by

giving it the name of U: XYZ or S: XYZ.

13.4.2 CAE DECALS DATABASE

A CAE Decal is the graphical representation of a "gate" in the ***PADS-LOGIC*** software. CAE Decals are normally created and edited in the Part Library routine of ***PADS-LOGIC***.

13.5 PART TYPE DATABASE

The PART TYPE Database contains the Electrical data for each different part in the database. The ELECTRICAL data is described in Section 10.1.1. Each different part is entered into the Part Type database and is cross referenced to its associated PCB Part Decal by a pointer, and the CAE Decal contained in the Part Type.

13.6 USE OF THE PLIB EDITOR

Creating and editing of all four databases is best done within the appropriate routines within ***PADS-LOGIC*** or ***PADS-PCB***. PLIB is used to perform other actions such as deleting obsolete parts or decals, listing parts, importing or exporting parts and decals using ASCII files.

NOTE: The EDIT feature of PLIB permits both the entering and changing of the alphanumeric content of the four databases.

When the PLIB program is called up, the Editor Message Window displays:

SELECT LIBRARY

This message is awaiting the selection of either the User Library (U) area or the Standard Library (S) area. Upon the selection of the library area, you are invited to select a database with the information line which reads:

DATABASES: LINES, DECAL, CAE DECAL, PART TYPE, INITIALIZE, HELP, BYE

Selection is made by typing the first letter of the desired Database or activity:

L = Lines Database
D = Decal Database
C = CAE Decal Database
P = Part Type Database
I = Initialize
H = Help Display for PLIB
B = Bye leaves PLIB. It is followed by "ARE YOU SURE" to confirm your intention.

13.7 P (PART TYPE LIBRARY)

When P is selected for the Part Type Database, the following occurs:

1. The Library area (User or Standard) and the Library name "Part Type" is displayed.

2. The number of Part Types in the Part Type Database is entered in the ITEMS indicator.

3. The ITEM NAME indicator awaits the entry of a Part Type name, after which the Part Type name will be displayed.

4. The Message Window requests the next action with the message:

SELECT ACTION

from the "Action" activity line.

5. The DATABASE Line is replaced with the ACTION Activity Line as follows:

FIND, NEXT, PREVIOUS, SAVE, DELETE, EDIT, WRITE, READ, LIST, HELP, BYE

Find: Use F to find or search for a Part Type. If the name of the Part Type is known, after F for find enter the part name <CR> and the Part Type Database for this named part will be displayed in the EDIT Window.

If the name of the Part Type is unknown, enter F followed by *<CR> and the first named Part Type in the Part Type Database will

be displayed. If you know the Part is within the 7400 series, enter:

74*

and a listing of all 74 Part Types will be made, starting with the first entry.

Next: N steps the currently displayed Part Type forward to the next Part Type in the Part Type Database.

Previous: P steps the currently displayed Part Type backward to the previous Part Type.

Save: After editing a Part Type, S is used to write the Part Type file to the Part Type Database LIB in PLIB.

Delete: D is used to delete a Part Type from the Part Type Database. D results in the message:

INPUT ITEM NAME

After the name is input, you are asked "ARE YOU SURE. Y/N." A "Y" will delete the item.

NOTE: The user is advised when deleting large amounts of data from the library to "Write" out to an ASCII file, delete the data from the ASCII file, then "Read" the file back into the library (after initialization).

Edit: Use E to enter the System Part Library Editor. A flashing "—" cursor appears in the Edit Window at the beginning of the first line. Strings of text can be entered as with conventional typewriter action.

The text of an existing file can be changed with the editor. The Character cursor is moved with the right, left, up and down "arrow" keys and the carriage return key, from line to line and within a line. Deletion of existing text is done using the Backspace Key.

Upon completion of editing, Escape and S for Save will save the file

under the PART TYPE NAME entered as the first entry made in the data format for a Part Type. While being "saved," format checks are made, and if improper entries have been made, an appropriate error message is displayed in the Message Window, and the line in error will highlight in red. The error must be corrected before the file can be saved.

The ESC key will abort the file and leave the Editor. The edited file can be SAVED or abandoned.

Merge (Part Type): M will allow the merging of the Part Type and Decal of a part from a ***PADS-PCB*** job file into the User Library, as could be the case, should you have access to the Database of a PCB design containing a part which you do not have in your PLIB User library and wish to copy the part into your User Library.

Select "M." After selecting M, the "Item Name" Indicator will flash. In response, enter the name of the Job File containing the part you wish to extract. You are then invited to enter the "Part Type Name" of the part you wish to extract. Enter the part type name followed by <CR>.

If the part type exists, a "Part Type Exists" message will result, and you will be prompted to Overwrite Y/N?.

If the part type does not exist, it will NOT be extracted, and the next "Part Type Name" message will be presented.

You should be aware that when a part type from a job is merged with the user library, the DECAL is also brought into the PLIB Decal Database.

Write: W will write out the item(s) you specify in an ASCII format. The "Write" feature's main purpose is for transferring individual library files from one user's system to another. The procedure is as follows:

Select "W". After entering W, the prompt is displayed:

ENTER OUTPUT FILE NAME (DIRECTORY USED)

It is awaiting input of a file name (8 characters or less) of where you would like the ASCII data to go.

NOTE: After responding with a name, the system automatically assigns an extension as follows:

.P	for part type
.D	for PCB decals
.L	for lines
.C	for CAE decals

depending upon which library you are writing from. The file can be displayed and edited by entering the Subdirectory and using PEDIT and the filename and extension.

After inputting the file name the following prompt appears:

ENTER ITEM NAME (*=ALL; EG: 74*=ALL ITEMS STARTING WITH 74

It is now awaiting input of the name of the library data you wish to write out. If the user types an "*", everything in that library will be written into the ASCII file.

If the file needs to be edited for some reason, the procedure is as follows:

Exit PLIB, and using a Word Processor, obtain the file to be edited as follows:

PEDIT \PADS\LIB\FILENAME.EXT <CR>

When saving the file, type \PADS\LIB\FILENAME.EXT <CR>

Read: Reads the files created by the write function above, back into the sublibrary. When selected, the message:

ENTER INPUT FILE NAME

requests the name of the file to be read in. Take care to ensure that Part Type files are read into the Part Type sublibrary. Lines are read

into the LINES sublibrary, etc.

Note that if an item already exists in the database, you will be asked to Overwrite Y/N?

List: L will print out a listing of the different Part Type names as contained in the Part Type Database. If a printer is unavailable, the data will go to the file PLIB.ERR.

A "*" will print out a directory of the Parts Type Library.

Help: H will display a "Help" file at the CRT summarizing the format of the sublibrary you are currently in.

Bye: B will exit the Part Type Database and enter the next higher level, i.e. System Part Library.

13.8 INITIALIZE

USE CAUTION WHEN SELECTING INITIALIZE. THIS FEATURE CAN BE USED TO CLEAR OUT OR ERASE THE CONTENTS OF THE ENTIRE STANDARD AREA OR USER AREA, INCLUDING LINES, DECALS AND PART TYPES.

To use, select INITIALIZE, and confirm your desire to initialize PLIB. All items in the four databases of the area selected will be erased, leaving this area of PLIB EMPTY. Therefore, the User is advised to "Write" each library out to an ASCII file, or create backup copies of the binary library files, prior to initialization (refer to 13.10.2).

13.9 SYSTEM PARTS LIBRARY

A comprehensive selection of Line Items, Parts and CAE Decals and Part Types are included with the ***PADS-LOGIC*** Program. These are included in the Standard Part area. A Print Out of the Directory of these items can be obtained by selecting the Standard area, using the "List Directory" feature (L) and responding with * for ALL, as outlined in Section 13.7 above or by reference to Appendix A.

13.10 MAINTENANCE OF SYSTEM PARTS LIBRARY

13.10.1 CREATING OR MODIFYING PART DECALS/TYPES:

The System Parts Library which includes lines, part decals, CAE DECALS, and part types with your system should be kept on the hard disk. At various times you will need to add a new part decal or part type to the library. Use the PARTS LIBRARY routine. The new part decal will be automatically placed in the User area of the library. When you wish to MODIFY a Part or its Decal stored in the Standard area, the SAVE function will save the modified item in the Standard area unless the S:XXX name is replaced with U:XXX. In the later instance, the part decal will be stored in the User area with the name XXX.

In the instance of Part Types, it is recommended that NEW Part Types created by the user, be created and stored in the User area. Standard area Part Types, originally supplied by CAD Software, Inc., which are modified by the user, should be modified and stored in the Standard area of PLIB.

13.10.2 BACKUP COPIES

We wish to emphasize that you should maintain a current backup copy of all Part Decal and Part Types on separate media from the hard disk. PERIODICALLY, (AT LEAST ONCE PER WEEK), YOU SHOULD FIRST CREATE A BACKUP FILE ON FLOPPY DISK IN THE EVENT A PROBLEM ARISES DURING UPDATING.

Note that a Part Type, within the System Parts Library, is not a file. As a result, individual library items cannot be transferred to and from disk storage as individual library item transfers. The entire System Parts Library file must be copied for storage rather than on a record-by-record basis. Similarly the entire Library file is read from floppy disk to hard disk for usage. In the process, any records already in the hard disk Library are over written by the file from floppy disk. Be sure you take this into consideration in planning your Library updating effort.

13.10.2.1 USER LIBRARY

The User Library consists of the following eight files:

UPART.DAT
UPART.IDX
UDECAL.DAT
UDECAL.IDX
UCAEDEC.DAT
UCAEDEC.IDX
ULINE.DATA
ULINE.IDX

All of these files can be copied to a high density floppy in one operation as follows:

Place a floppy in the destination drive (in this example, drive A:), enter the subdirectory \PADS\LIB by entering the command:

CD \PADS\LIB <CR>

At the prompt:

C: \PADS\LIB>

you enter: COPY U*.* A: <CR>

To restore the user library to the hard disk from floppy, enter the subdirectory \PADS\LIB by CD \PADS\LIB <CR>

At the prompt:

C: \PADS\LIB>

you enter: COPY A:U*.* <CR>

This will copy the entire User Library back to the subdirectory \PADS\LIB which will cause the current contents of the User Library to be overwritten.

When archiving Decal information, it is necessary to include the LINE files as well. Although it is not apparent to the user, some of the decal information is stored in the Lines library.

If the User Library is too big to be stored on one floppy, you may store some of the files on one floppy and the rest on additional floppies. The DOS utility BACKUP provides a means to copy files onto multiple floppys (refer to your DOS Manual for details).

CAUTION: Again, we caution you that before updating any of your System Sub Libraries, ALWAYS save the Library contents on floppy, as insurance against potential problems.

13.10.2.2 STANDARD LIBRARY

The Standard Library consists of the files:

SPART.DAT	SPART.IDX
SDECAL.DAT	SDECAL.IDX
SCAEDEC.DAT	SCAEDEC.IDX
SLINE.DAT	SLINE.IDX

If you make substantial modifications to the Standard Library, it should be backed up periodically. The Standard Library is too big to copy on a single high density floppy, hence the DOS BACKUP utility or an equivalent "multi-volume" backup utility must be used.

13.11 A SUMMARY OF THE *PADS-LOGIC* LIBRARY ORGANIZATION

The creation of Part Types, Part Decals, their storage in the Library and use in a Schematic or Board database is a highly sophisticated sequence involving a number of different routines in the PADS software. A description of each of these routines is provided in complete detail in their dedicated chapters in this Reference Manual. However, their integration into an integrated system is not covered in the respective chapters. This section (13.11) carries the discussion one step further by providing a road map of the interaction and integration of the various routines that comprise the total PADS Library system.

Section I - A SUMMARY - The PADS LIBRARY

Refer to the figure below for a discussion of the PADS Library:

PLIB	
USER Area	STANDARD Area
2-D Lines	2-D Lines
Part Types (Electrical)	Part Types (Electrical)
CAE Decals	CAE Decals
PCB Decals	PCB Decals

1. The PADS Library is controlled by the Library Management program called PLIB. To access the library, be in the Root directory and in response to C: > respond PLIB <CR>. The response is the display at the CRT of the PLIB Editor, of an alphanumeric text editor (see Section 13.2 PLIB).

2. The PADS PLIB Library contains two areas, the Users area and the Standard area.

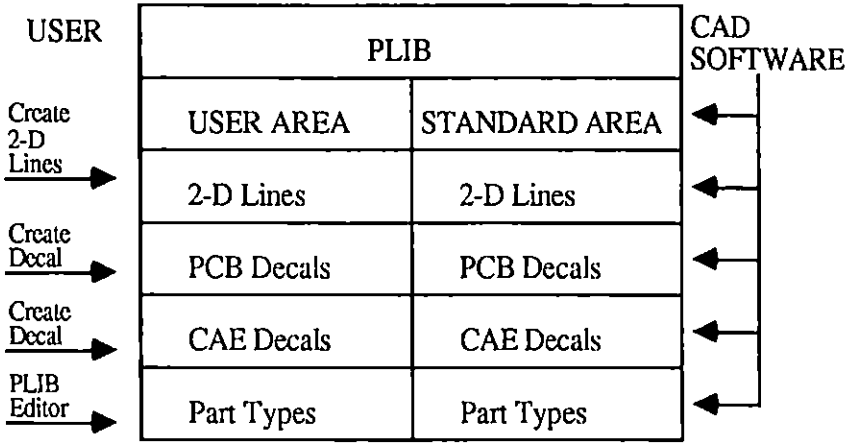
3. The Users area contains those items that have been created by YOU, the user. The Standard area contains those items supplied by CAD Software, Inc.

4. The User area contains four categories of items in four sub-libraries: 2-D Lines, CAE Decals, PCB Decals and Part Types (Electrical).

5. The Standard area contains four categories of items in four sub-libraries: 2-D Lines, PCB Decals, CAE Decals and Part Types.

Section II - PLIB CONTENTS GENERATION

Refer to the following PLIB Generation figure during the reading of this section:



1. 2-D Lines, PCB Decals, CAE Decals, and Part Types are stored in the User and Standard areas of PLIB, however, their original generation is from a variety of sources:

2. The 2-D Lines data contained in the Users area is created by the user using the 2-D LINES Routine. The creation is a graphical function using the 2-D Line Graphical Editor, however, the stored item is textual content, and stored in the PLIB 2-D Line Users area.

3. PCB Decals are created as graphical entries in the ***PADS-PCB*** PART DECAL graphical Editor and normally stored as Text data in the PCB Decal Sub-library of the Users area (assuming you have the ***PADS-PCB*** Board CAD System).

4. CAE Decals and Electrical data are created in the ***PADS-LOGIC*** Capture system, using the PART LIB Routine.

NOTE: While it is possible to use the PLIB Text Editor to modify the Text data associated with 2-D Lines and Decals, this is not recommended. Instead, it is strongly recommended that the 2-D Line and Decal Graphical Editor always be used.

5. The 2-D Lines, Part Decals and Part Types stored in the Standard area of the PLIB Library have been created by CAD Software, Inc., the ***PADS-LOGIC*** software vendor.

6. To modify a CAD Software, Inc., supplied 2-D Line or Part Decal stored in the Standard area, the user uses 2-D LINE or PART LIB and modifies these software vendor created items. After modification by the user, the user must save the modified item; in turn, it is automatically strored in its original storage area, i.e., the Standard area, unless changed by the user. To store in the User area, SAVE using the name U:XXX where U indicates "to be stored in the User area" and XXX is the name of the item.

7. The user may also modify the Part data supplied by the software vendor and stored in the Standard area. To do so, the user edits the Text data with the PART LIB Routine and saves it in the Standard Part Type area.

Chapter 14

PARTS LIBRARY

14.0 INTRODUCTION

This chapter of the ***PADS-LOGIC*** User Manual describes the creating and modifying of Parts that are included in the ***PADS-LOGIC*** software.

Included with the ***PADS-LOGIC*** program are over 3,000 Parts, ready for your use. These Parts are listed in alphanumeric sequence in Appendix A. Hardcopy plots of the Decal are contained in Appendix B.

Included in ***PADS-LOGIC***, is a Parts Library Editor that allows the generation of additional parts as well as the modification of existing parts. The use of this Parts Library Editor is the subject of this chapter of the ***PADS-LOGIC*** User Manual.

The Parts Library routines are contained in F9 HIER/LIB routine of the Main routine. The selection of F9 HIER/LIB will result in the following menu window:

OTHER MENU
HIER/LIB
1 PART TYPE
2 CONNECTOR
3 OFF PAGE
4 REP OFF PAGE
5 GATE DECAL
6 PIN DECAL
7 HIERARCHY (Chapter 15)
8
9
0 EXIT

The following provides a quick-look summary performed by each of the subroutines. Following the quick-look summary, each of the subroutines will be described in detail.

PART TYPE is used to create an entirely new part or modify an existing part which is saved in the Part Library.

NOTE: Where possible, it is strongly recommended that a new part be created by selecting an existing part to modify, and then saving the modified part under a name that is associated with the new modified part.

CONNECTOR is used to create and modify Connector Parts. Connectors are normally shown at the Pin level on the Schematic rather than at the Part level, hence a different subroutine is used to create Connectors.

OFFPAGE is used to modify the symbols used for Off-Page, Power and Ground.

REP OFF PAGE is used to bulk replace Off-Page, Power and Ground symbols that are currently in a schematic with a new Off-Page, Power and Ground symbol that you have created.

NOTE: CAD Software has created several Off-Page, Power and Ground symbols.

GATE DECAL is used to create a new Decal or modify a Decal for use with a Library Part.

NOTE: A Decal is the graphical representation of a Part or its Gate.

PIN DECAL is used for creating the information associated with a new pin. There are a number of different pin types provided with the ***PADS-LOGIC*** software. The user can create new pins and add to this repertoire of pins.

EXIT is used to Exit from the Part Library Editor and return to the OTHER MENU routine.

The remainder of this chapter will describe each of the six subroutines of Part Library, in detail.

14.1 PART TYPE

Use PART TYPE to create or modify a new Part that is to be added to the existing Parts in the Part Library.

Where possible, and in the instance of saving time, if a Part exists in the Library that is reasonably similar to the New Part that is to be created, modify the existing part and save it under the New Part Name.

When PART TYPE is selected, respond to the prompt:

ENTER PART TYPE>

with the name of the Part that is to be created or modified. If the Part Type is a New Part and does not Exist, the following prompt will appear:

PART XXX DOES NOT EXIST - DO YOU WANT TO CREATE PART (Y/N?)

A response of Y will result in the menu below:

If editing an existing part, the software displays the "Gate" information at the CRT and the menu changes to the following:

OTHER MENU
HIER/LIB
PART TYPE
1 ELECTRICAL
2 MOD GATE
8 CHECK PART
9 SAVE PART
0 EXIT

Before discussing the details of this routine, let us distinguish between Electrical and Gate information.

The ELECTRICAL information contains the Electrical data for each different part in the database. The Electrical information consists of the Gate or Decal, Technology, Ground and Power Signal Pin

identification and other information of an administrative nature, such as price, manufacturer, etc. The "Electrical" Data for each different part is entered into the Part Type database and is cross referenced to its associated Gate information by a pointer, i.e., the Gate Decal Name contained in the the Part Electrical information.

GATE information, on the other hand, contains the Decal and Pin Data of the component as used to display the component at the CRT, or when plotting the schematic. This may be Multiple Gates such as with a Multiple-Gate Integrated Circuit or it may be just the Resistor Symbol in the instance of a Resistor.

Gate and Decal information is created in the F5 GATE DECAL subroutine. The graphical decal information is saved to the CAE Decal database of the user area in the system's Part Library. The Electrical pin information such as Pin Number, Pin Type and Swap class is created at the gate level but stored in the Part Type database. The same Decal can be used for many different parts. Parts Decal INV, for example, is an Inverter and can be used in those IC's containing Inverter Gates.

CAUTION: The same Decal may be used from Part to Part, however, if the Pin (Stub) is moved or changed, all Part Types using the Decal will change.

14.1.1 ELECTRICAL

F1 ELECTRICAL contains a Text Editor used to create the Electrical and Administrative information of a Part. When F1 ELECTRICAL is selected, the CRT changes to display the Text Editor. The Text Editor is preformatted with the following questions which are answered by selecting the Answer Line with the cursor and entering the applicable information.

PART TYPE OR SUFFIX
PART TYPE PREFIX LIST
LOGIC FAMILY
PCB DECAL AND ALTERNATES
NUMBER OF PART ATTRIBUTES
DISPLAYED ATTRIBUTE LINES
NON-NUMERIC PIN NUMBERS (YES/NO) LINES REQUIRED

NUMBER OF SIGNAL PINS
NUMBER OF GATES
TYPE GATE DECAL AND ALTERNATES

The following describes the information contained in each line of the Electrical Text Editor:

PART TYPE SUFFIX

For Integrated circuit parts, this line contains the last part of the Part Type Name, which when combined with the Prefix (below), serves to uniquely identify the Part.

The combination of the Part Type Prefix and Part Type Suffix uniquely identifies the Part. A Part Type can have, between the combination of the Suffix and Prefix, a total of 16 Alphanumeric characters. It may not have special characters, however. The following characters are illegal: *()[]{}:

Integrated circuits, especially TTL, may, in the interest of maximizing the contents of the Library without increasing the data handling requirements, use a combination of Prefix and Suffix. The Suffix contains a form of shorthand that allows an entire family of IC's to be stored under one Part Suffix. The first part of the Suffix of the Part Type Name will be a backslash thus \ . The numbers following the \ are those used to uniquely identify the device family: 00 for a 7400, 192 for a 54192, etc.

PREFIX

The Part Type Prefix in combination with the Suffix is used for TTL Parts of the 54 and 74 series, so as to include all variations of the family without an individual Part entry in the library for each variation. The first entry in the Suffix Data Line is used to indicate whether the Part is used for both 54 and 74 variations or one or the other. If both 74 and 54 are used, enter X4. If only 54 is used, enter 54; if only 74 is used, enter 74.

Next is listed the variations of the device, i.e., ALS, H, L, etc. If both the 54 and 74 series has the variation of ALS, use X4 and enter X4ALS (not 54ALS and 74ALS). If only the 54 series has a

variation of L, for example, enter 54L (but not X4L or 74L).

The typical entry for the 5404 and 7404 series of TTL components would, for the Prefix List, read X4 X4AS X4AH 54L X4LS X4S X4HC, while the Suffix Line would read \04. Non-TTL Parts are entered by the Vendor's data sheet device number, i.e., 68000, 68020, etc.

Discrete Parts, because of their potentially large family of items in the Part Library, use a "family and attribute" value convention. The Part Type Name for a Discrete Part can be R1/4W or R2W, etc., and the Value is assigned AFTER the Resistor is brought into the schematic.

LOGIC FAMILY

This line contains the family with which the Part is associated, i.e., TTL, CMOS, ANA (for Analog), or UND (for unidentified). The User may choose any three characters to classify the family.

Contained in the ***PADS-LOGIC*** Files is a file called PKGTYP.DAT. This file associates a REF DES Prefix to each Logic Family, i.e., U for IC's, R for Resistors, etc. To obtain a copy, or edit the file, be in the Root Directory and use PEDIT or Print.

NUMBER OF PART ATTRIBUTES

A maximum of ten attributes can be associated with the Part. Generally, if used, these Attributes should be used as follows:

PART DESCRIPTION
COST
VENDOR PART NUMBER (OR VALUE FOR DISCRETE PART)
MANUFACTURER NUMBER 1
MANUFACTURER NUMBER 2 (OR TOLERANCE OF PART)

The User is free to create new attributes such as "Vertical Height" which might be useful for mechanical packaging or thermal analysis.

DISPLAYED ATTRIBUTE LINE

The two entries following, permit the display of Attribute Lines at the CRT and on the Plot. A maximum of two Attribute Lines can be displayed at the CRT and plotted, depending upon if requested or not. In the First entry, place the Line Number of the First Attribute to be displayed; place the Number of the Second Attribute in the Second entry.

NON-NUMERIC PIN NUMBER (Y/N?)

If the part uses Alphanumeric Pin Numbers, i.e., A1, A2, etc., respond Y and Data lines will be available (according to how many are specified in the Lines Required slot) to create an association table between these Alphanumeric Numbers and Numeric Pin Numbers.

NUMBER OF SIGNAL PINS

The system has the ability to generate Power and Ground Connection Nets automatically if this feature is used on Parts with Standard Power and Ground. If this feature is used in the Part, it is not necessary to show Power and Ground on the Gate or Part on a sheet of the Schematic. Instead, when a Part is used in a sheet of the schematic that has its Signal Pins identified in the Library, the system automatically extracts the Power and Ground Pins and creates Power and Ground Nets which include those parts. Enter the Power and Ground pin numbers for the Signal Pins that have Standard Power and Ground.

For both Power and Ground, enter the Pin Number under the column, PIN NUMBER. Enter the Signal Net Name under Signal Name, and indicate the PCB Track Width desired (in mils). In ***PADS-LOGIC***, Power and Ground for DIP Parts is 50 mils, for SMD Parts, the Width is the width of the SMD Decal pad. The latter is used to assign a Track Width to the indicated Signal Net in the Schematic Net List that can be used for those Printed Circuit Board CAD systems that can utilize Track Width in the Connection List.

NOTE: In ***PADS-LOGIC*** Parts, Ground is called GND; Power is called +5V.

NUMBER OF GATES

Enter the number of Gates that are associated with the Part. A 74LS04, for example, would have six gates. A Part that is displayed at the Component Level would have the number 1; for example, a 74ALS365 would have 1. Discrete Parts, such as Capacitors and Resistors that are shown at the Component level, would use a 1.

TYPE (SWAPPABILITY)

For each Part Gate, enter the Swap Type, Primary Gate Decal Name and the Alternate Decal Name.

The Type is used to indicate whether the Gate is swappable with other Gates within the same Part or other Parts of the same type in the schematic: a zero indicates the gate is not swappable, a 1 indicates the gate is swappable with gates of the same Part Type on the board. If a part contains more than one type of swappable gate, then identify the second type with the number 2, etc.

GATE DECAL AND ALTERNATE

Enter the Primary Gate Decal Name, followed by a space, followed by the Alternate Decal Name or DeMorgan Equivalent Decal Name. Up to three Alternate Decals may be specified.

EXAMPLE

The following is an example of the Electrical format for a 74LS04.

PART TYPE SUFFIX
Response: \04

PART TYPE PREFIX
Response: X4 X4AS X4H 54L X4LS X4S 4HC X4HCU

LOGIC FAMILY
Response: TTL

PCB DECAL AND ALTERNATE
Response: DIP14 DIP14\SO

NUMBER OF PART ATTRIBUTES
Response: 5

PART DESCRIPTION: HEX INVERTER
DOLLARS: $
VENDOR NUMBER:
MANUFACTURER #1: TEXAS INSTRUMENTS
MANUFACTURER #2:

NOTE: For Discrete Parts, the Third Attribute Line is used for VALUES to be assigned to the Part. For example, on a Resistor or Capacitor, enter VALUE: ??? on the Third Attribute Line. Then ensure the Third Attribute Line is displayed. Save the Part and when used on the Schematic, the ??? can be changed to 10K, 1UF, etc. Please be advised that if porting a ***PADS-LOGIC*** Net List to ***PADS-PCB***, the combined Part Type and VALUE may not exceed 15 characters.

Use the fifth Attribute Line for Part Tolerance by entering: TOLERANCE ??? Save the part and when used, the Part Tolerance can be set correctly for each part.

DISPLAYED ATTRIBUTE LINES:
Response: 0 0

NON-NUMERIC PIN NUMBERS Y/N LINES REQUIRED
Response: N 0

NUMBER OF SIGNAL PINS
Response: 2

PIN NUMBERS	SIGNAL NAME	PCB TRACK WIDTH
RESPONSE: 7	Response: GND	Response: 50
14	+5V	50

NUMBER OF GATES
Response: 6

TYPE	GATE DECAL AND ALTERNATE	
1	INV	INVDM
1	INV	INVDM
1	INV	INVDM
1	INV	INVDM
1	INV	INVDM
1	INV	INVDM

14.1.2 MOD GATE

The MOD GATE routine is used to create or modify the Gate associated with the Part. Between F1 ELECTRICAL and F2 MOD GATE, all activities associated with creating a Part or modifying a Part can be accomplished from the Part Library Editor.

When MOD GATE is selected for a new part, the following prompt is displayed:

CREATE BOX SYMBOL AUTOMATICALLY (Y/N?)

In addition, at the CRT is displayed four lines of Text information contained in the Text Block.

Line Number 1: This line reads U999 and is the default space reserved for the true Reference Designation for the Part when brought up on the schematic.

Line Number 2 - Part Name: This line contains the Part Name of the new Part and is Text Space Reserved for Part Name when brought up at the CRT.

Line Number 3 - ATTR1: This is one of two possible attributes of Text created in the Electrical section of the Part that has been selected for display with the Gate.

Line Number 4 - ATTR2: This is the second possible attribute of Text that has been selected for display with the Gate.

If the response to the prompt:

CREATE BOX SYMBOL AUTOMATICALLY (Y/N?)

is Yes, another prompt asks you to:

ENTER NUMBER OF INPUT AND OUTPUT PINS>

In response, enter first the number of Input pins for the Gate, a space, followed by the number of Output pins <CR>

A red pop-up window appears with several pin type Names and Decals contained therein, together with the prompt of:

SELECT PIN DECAL FOR INPUT PINS

With the cursor, select the Name of the Pin Decal that you wish to use for the Input Pins. The selection of the Decal Input Pins is followed by a prompt that reads:

SELECT PIN DECAL FOR OUTPUT PINS

Your response, using the cursor, is to select the Pin Decal for use with the Output Pin.

NOTE: These Pin Decals are for your convenience in imparting Graphical information to the Decal. The different Pin Types are NOT used as data in any Data Checking activities. You may create your own Pin Decals and Symbols if you wish.

The CRT is redrawn with a Rectangle Gate Symbol created automatically, with the Input Pins on the left side of the Part Gate Decal and the Output Pins on the right side of the Gate Decal. The menu changes to the following:

OTHER MENU
HIER/LIB
1 2-D LINES
2 ADD TEXT
3 TERMINALS
4 SETUP
5 COPY
6 MOVE
7 DELETE
8 ORIGIN

9 COMPLETE
0 EXIT

14.1.2.1 2-D LINES

This subroutine is used to modify the Gate Decal Outline. The Gate Decal Outline is comprised of 2-D Lines. To use, select F1 2-D LINE and the menu will change to the following:

OTHER MENU
HIER/LIB
2-D LINES
1 NEW POLY
2 NEW CIRCLE
3 MODIFY
4 COMBINE
5 COPY
6 MOVE
7 DELETE
8 FROM LIB
9 TO LIB
0 EXIT

These subroutines are the same as used for 2-D LINES in the Schematic program.

NOTE 1: Use 2-D LINES to change or modify the automatically created symbol for the Decal to something that is more appropriate for the Gate.

NOTE 2: When the sides of the Gate Decal Outline are moved, it is necessary to relocate the Pin Decal with respect to the Pin Decals and other Text data relative to the new Gate Decal Outline.

NOTE 3: FROM LIB and TO LIB do NOT, repeat, do NOT apply to the Part Type Library. Instead, these routines allow you to save to the 2-D Line Library, any 2-D Line Items and add them to the 2-D Outline Decal presently displayed.

14.1.2.2 ADD TEXT

This routine allows the adding of free text to the Part Decal. Note that when added, and subsequently displayed at the CRT or plotted, it is created as Free Text even though it is part of a Decal.

14.1.2.3 TERMINAL

This routine allows complete manipulation, modification and addition of Terminals to the Gate Decal. When selected, the Menu window changes to the following:

OTHER MENU
HIER/LIB
TERMINALS
1 ADD TERMINAL
2 CHG DECAL
3 CHG SEQ NO
4 SET PNUM
5 SET PNAME
6 SET PTYPE
7 SET PSWAP
8 CHG PNUM
9 CHG PNAME
0 EXIT

14.1.2.3.1 ADD TERMINAL

The use of the routine ADD TERMINAL allows the addition of Terminals above and beyond those specified when the Gate was automatically created.

Select F1 ADD TERMINAL and the Red Pin Symbol pop-up window appears. Select the Pin Decal as appropriate using the cursor. The appropriate Decal is attached to the cursor. While attached to the cursor, you should complete all the Text Data associated with the Pin, including the Pin Number, Pin Name, Pin Type and Pin Swappability, as well as to change the Pin Decal; if necessary, Rotate it, Mirror it in X and Y, Move the pin to the appropriate place on the symbol, and finish with F1 COMPLETE. Upon COMPLETE, the Pin is placed in its location and another

identical Pin is created, attached to the cursor for use in assigning another Pin. The right mouse button (FO EXIT) terminates the sequence.

14.1.2.3.2 CHG DECAL

CHANGE DECAL permits a Pin Decal already on the Schematic to be replaced with a different choice from the Red PIN SYMBOL pop-up window.

To use, select F2 CHG DECAL. Then SELECT the New Pin Decal from the pop-up window, then select the Pin (or Pins) on the Gate that are to be replaced with the New Pin Symbol.

14.1.2.3.3 CHG SEQ NO

PIN SEQUENCE NUMBER is the first Number in the Pin Text Data (#N) and is used to indicate the sequence in which Pins are added to the Decal. It has no relationship to the Pin Number. CHG SEQ NO is used in those instances where, within a family of parts, the pinning of terminals of different parts are different, such as is the instance of a DIP and SMD version of the same part.

To change the Sequence Number, select F3 CHG SEQ and respond to the prompt by selecting (with cursor) the Sequence Number (#X) with the cursor. Respond with its NEW Sequence Number and will be swapped with the Pin currently using the New Sequence Number.

14.1.2.3.4 SET PNUM

SET PIN NUMBER is used to set the Electrical Pin Number on the Gate. When selected, respond to the prompt:

ENTER ELECTRICAL PIN NUMBER (CR=1)>

Enter the number and select with the cursor the pin to use that number.

Suggestion: The Set Pin Number has an Automatic incrementing function, therefore, start with Pin One and each succeeding selection

will increment the Pin Number by one. Where Power and Ground Pins are NOT to be shown on the Decal, keep cursor positioned on a pin and "run by" those pins, by clicking beyond them.

14.1.2.3.5 SET PNAME

SET PIN NAME is used to assign a Pin Name (Not its Pad Number) to a Pin. To use, select F5 SET PNAME and respond to the prompt:

ENTER PIN NAME (CR=NONE)>

with the Pin Name <CR>. Then place the cursor over the Pin to be Named and use F1 SELECT. The Pin Name will be displayed inside the Gate Decal next to the Pin. Set Pin Name has an automatic incrementing function also; as each pin is selected, the name is incremented. This is extremely useful for address and data pins.

14.1.2.3.6 SET PTYPE

PIN TYPE defines the functions of the Pin on a Part as used in the circuit. Its usage is in the REPORTS routine to provide a "Status Report" on the Logic of the Design.

To set a Pin Type, be in MOD GATE, TERMINALS, select F6 SET PTYPE and respond to the prompt:

ENTER PIN TYPE (CR=U)>

with the desired Pin Type. Then select the Pin whose Pin Type is to be set, and observe that TYPE = X is changed to the New Pin Type.

The legal pin types are as follows:

S = Source Pin
B = Bidirectional Pin
C = Open Collector Pin
O = Or-Tieable Source Pin
T = Tri-State Pin
L = Load Pin

Z = Terminator Pin
P = Power Pin
G = Ground Pin
U = Undefined

14.1.2.3.7 SET PSWAP

Setting of PIN SWAP establishes the Swappability of a Pin on a Component when commanded to be swapped from a back annotative file generated in the Board CAD system. If the Board CAD System used with ***PADS-LOGIC*** is ***PADS-PCB***, a routine called "PARTS PLACEMENT" swaps the Pins on all "Unfixed Swappable Pins." If swapping of the pins results in shortening of the total connection length on the board, the results of this swapping are contained in an ECO File sent from the ***PADS-PCB*** Board System to ***PADS-LOGIC***.

If the Pins have been defined in Set PIN SWAP, they will be swapped in the schematic in ***PADS-LOGIC***. To use, be in MOD GATE, TERMINALS and select F7 SET PSWAP. Respond to the prompt:

ENTER PIN SWAP CLASS (CR=0)>

with the Class, then select the Pin whose swappability is to be set. Observe the SWP = X is changed to the desired class.

The Legal classes are:

0 = Not Swappable under any circumstances
1-9 = The Pin can be swapped with another pin with the same numbered class, within the same gate of a part. (Normally, use a 1 to indicate swappability.)

14.1.2.3.8 CHG PNUM

CHANGE PIN NUMBER is used to change the number of an existing pin on a Decal. To use, place the cursor over the pin and select F8 CHG PNUM and respond to the prompt with the new Pin Number. This routine has one less button or key stroke than SET PNUM when changing just one pin.

14.1.2.3.9 CHG PNAME

CHANGE PNAME is used to change the Pin Name of a pin already on the Decal. To use, place the cursor over the pin to be changed and respond to the prompt with the new Pin Name. The Pin Name will be changed to the New Name. As with CHG PNUM, one less key stroke is required than SET PNAME when changing just one pin.

14.1.2.4 SETUP

F1 DISPLAY - This is used the same way as at the schematic level except that the item categories are appropriate for editing parts instead of sheets.

F2 PARAMETERS - This routine contains a subset of the parameters at the schematic level. It operates identically to the schematic level PARAMETERS routine.

F3 MACROS - This routine is used the same way as at the schematic level.

F4 NAME DISP(LAY) - This routine is used to toggle On and Off the eight various types of Text associated with a Gate Decal. You select F4 NAME DISPLAY and a Red NAME DISPLAY pop-up window appears with eight Text Types displayed. The Default position is in the On or Displayed position. To toggle a Text Type to Off, place the cursor over the Text Type and select F1 and the Text Type will turn Blue, indicating it has been picked and the toggle will change to Off.

NOTE: Text Display Toggle is used only for toggling On or Off the Text at the CRT, as well as whether the Text will be plotted with the Schematic.

14.1.2.5 COPY

COPY is normally used to copy a terminal.

COPY can be used to duplicate the Gate Outline and thus have two Gate Decal Outlines in the Gate Decal. REF DES, Part Type and

Attribute Text may not be copied.

To use, place the cursor over the item to be copied and select F5 COPY.

NOTE: Obviously this feature is used in rare special instances and only where a Gate Decal is intended to have two Gate Decal Outlines.

14.1.2.6 **MOVE**

MOVE can be used to Move any items associated with a Gate Decal, including Attributes. To use, place the cursor over the item to be moved and select the item. The item is attached to the cursor and the menu changes to allow further manipulation of the item while in the process of being moved. The manipulation menu presented during MOVE is a function of the item selected. For example, moving of Text allows one to Rotate, Change Size, Center, Left and Right Justify, both Horizontally and Vertically.

14.1.2.7 **DELETE**

DELETE allows the deletion of Decal Outlines and Pins associated with a Decal. To use, place the cursor over the item to be deleted and select F7 DELETE.

14.1.2.8 **ORIGIN**

ORIGIN is used to relocate the origin associated with a Gate. When a Decal is automatically created, the origin is automatically placed at the lower left hand corner of the Gate. The origin is the point by which the Gate is referenced while being brought into the schematic and placed, rotated, etc. To use, place the Origin in the new location and select F8 ORIGIN. The Origin symbol will be relocated to the location of the cursor and all points and items on the Part Decal will be automatically reassociated with the new Origin location.

On Off-Page, Power and Ground Symbols, and Connectors, the Origin is the point on the Symbol to which the connection is attached (there being No pins on these four symbols).

14.1.2.9 COMPLETE

When you have finished editing at the gate level, use this routine to return to the part level and preserve your edits.

14.1.3 CHECK PART

CHECK PART is used to check the correctness of the Decal and the Part in accordance with the structure of the format set up when creating the Electrical Part Type data.

NOTE: CHECK does not check for the technical accuracy of the Part against the Vendor's technical specification. Instead, it is a structural check to insure that, if you have indicated your intention for an item, an omission has not been made during the creation of the Part. For example, while creating the Electrical Part Type data--you indicated there would be six gates. A check is made to see that indeed there are six lines of Gate-type data. If not, an error indication will be generated. Use CHECK! It will save you much grief in creating your Library Parts.

CHECK PART will inform you on the Red message line of the number of errors found. Use the ALT 0 Key, if you have no printer, to display the errors in the errors file.

14.1.4 SAVE PART

After CHECK PART is run and error free, SAVE PART is used to save the part that has been created in the User's Library. When selected, the following prompt will be displayed.

ENTER PART TYPE TO SAVE (CR = U:XX)

If you wish to save the Part in the User's Library, utilizing the same name used as to create the Part, then respond <CR>. If, on the other hand, you wish to change the name before saving, enter the new name.

NOTE: By default, a Part created by the user is always saved to the User sub-library of the Part Library. Parts shipped with the system are stored in the Standard Sub-Library of the Part Library.

Note that in front of each listed library part is either a U: or an S: This indicates which of the two sub-libraries of the Part Library the Part is located. U indicates the Part has been created by the User and stored in the User sub-Library; an S indicates that the Part was created by CAD Software and stored in the Standard sub-Library of the Part Library.

14.1.5 NON-AUTOMATIC CREATION OF A GATE DECAL

When creating a new Gate Decal and the decision is made NOT to automatically create the Gate Decal Symbol and the response to the prompt:

CREATE BOX SYMBOL AUTOMATICALLY (Y/N?)

is NO, because you wish to create the Decal yourself or modify an existing Decal, the system does NOT create the Decal Outline; instead, the CRT presents the four Text items and the menu changes to the following:

OTHER MENU
HIER/LIB
1 2-D LINES
2 ADD TEXT
3 TERMINALS
4 NAME DISP
5 COPY
6 MOVE
7 DELETE
8 ORIGIN
9 COMPLETE
0 EXIT

Creation of the Gate Decal under these circumstances is identical to when the decision was made to Automatically generate the Part Decal Outline except for two instances:

1. 2-D Lines must be used to create the Part Decal Outline, after which the four Text items are located correctly with respect to the newly created Outline, and those Text items NOT to be displayed are set to OFF.

2. The Origin should be located correctly with respect to the Part Decal Outline (usually the lower left corner of a Gate and the center of a Resistor or Capacitor).

3. Use the ADD TERMINAL routine to create the pins for the gate.

4. Complete the Decal from this point in the same manner as if the symbol were automatically created. Check and Save.

14.1.6 MODIFYING EXISTING PARTS AND GATES

PART TYPE is used to modify an existing part that has already been created and stored in the library.

To use, select F1 PART TYPE and respond to the prompt:

ENTER PART TYPE>

with the Name of the Part you wish to modify. If there are different versions of the Part, such as a DIP version and a pin grid array version or an SMD version, the names of these versions will appear in a pop-up window. Select the version of the Part that you wish using the cursor. To edit the gate pin information at the gate decal, select F2 MOD GATE.

Respond to the prompt:

ENTER GATE NUMBER TO EDIT (CR=1)

with the number of the gate that you will want to edit. Generally you will want to edit Gate Number 1, then move to Gate Number 2, etc. The Gate will be displayed along with the MOD GATE menu routine as follows:

MOD GATE
1 2-D LINES
2 ADD TEXT
3 TERMINALS
4 SETUP
5 COPY

6 MOVE
7 DELETE
8 ORIGIN
9 COMPLETE
0 EXIT

These subroutines are used in exactly the same fashion as were used in the construction of a New Part and will not be repeated here.

After you have made the modification to each Gate, use F9 COMPLETE.

Following the modification of the Gates, modify the Electrical portion of the Part if necessary.

NOTE: Always, repeat ALWAYS use F8 CHECK PART after you have modified a Part before saving the part.

NOTE: Use caution when moving terminals, changing Pin Types, etc. If a gate is used in more than one Part Type, all Part Types in the Library using that gate will be modified.

14.2 CONNECTOR

CONNECTOR is used for creating or modifying the Parts associated with a Connector (or Connector-like Parts).

A special routine is provided for creating a Connector Part because of its difference from other parts. Specifically, when a Connector is used on a schematic, one wishes the freedom of operating at the Pin Level, rather than the Complete Connector Part Level; secondly, one wants to use a Reference Designation, i.e., P1, and then have a Pin Number associated with the Reference Designation that is incremented each time one adds a Connector Pin to the Schematic (i.e., P1.1, P1.2, etc.); and finally, one does not necessarily need the Part Number associated with the Connector each time a connector pin is used. For these reasons, a special routine called CONNECTOR is provided. CONNECTOR can also be used for Wire Wrap Jumper Pins, Jacks, etc., that are similar in application to Connectors.

To use, select F2 CONNECTOR. A prompt asks:

ENTER PART TYPE>

Respond with the name to be used. The menu changes to the following:

OTHER MENU
HIER/LIB
1 ELECTRICAL
2 MOD GATE
8 CHECK PART
9 SAVE PART
0 EXIT

If the Connector does NOT exist, you will be asked if you wish to create it. If the connector exists, it will be brought to the CRT.

14.2.1 ELECTRICAL

When selected, a Text Editor customized for the creation of a Connector Part is presented. This Text Editor is used to create the Electrical information for a Connector as follows:

CONNECTOR PART TYPE
NUMBER OF CONNECTOR PINS
LOGIC FAMILY
PCB DECAL AND ALTERNATES
NUMBER OF PART ATTRIBUTES
DISPLAYED ATTRIBUTE LINES
NON-NUMERIC PIN NUMBERS
NUMBER OF CONN PIN TYPES
PTYPES CONNECTOR PIN DECAL

Complete the Connector Electrical data as follows:

CONNECTOR PART TYPE - This is the Part Type to be used to identify the Connector. This is the Part Type that will be contained in the Net List transferred to the Board CAD System and must be the same as used in the Board CAD system to identify the connector. Use some scheme to code the Connector. For example, CON/36P/ED would indicate a 36-Pin Edge Finger Connector.

NUMBER OF CONNECTOR PINS - Enter the total number of Pins contained on the Connector.

LOGIC FAMILY - Enter the Connector family to be associated with the Connector Part. Typical entries are CON, JAC, TP, etc.

PCB DECAL AND ALTERNATES - Enter the Decal Name used in the Printed Circuit Board software associated with the Part. If the Printed Circuit Board Part Library has Alternate Decals, enter these Alternates.

NOTE: The PCB DECAL and ALTERNATE is available in the Net List originating from ***PADS-LOGIC*** and can be used in any PCB software if the PCB software accepts Board Part Decal information from the Logic Capture system.

NUMBER OF PART ATTRIBUTES - Enter here the number of Part Attributes that will be displayed at the CRT and, if so, how many.

DISPLAYED ATTRIBUTE LINE - Indicate if the Attribute Lines are to be displayed at the CRT and, if so, indicate the number of the Lines to be displayed (2 maximum). Normally on a Connector Part, you will display only the REF DES.

NON-NUMERIC PIN NOS (Y/N?) - Use only if Pin Numbers are NOT Numeric, but rather Alphanumeric, i.e., A1, B6, etc.

NUMBER OF CONNECTOR PIN TYPES - Enter here the different number of Connector Pin Types. The software allows a different decal to be created for each pin type.

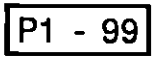
PTYPES CONNECTOR PIN DECAL - Enter here the Connector Pin Type and the Decal to be associated with the Connector Pin. Use a different line of data for each Pin Type. Typically, there will be three types: Input (Source), Output (Load) and Bi-directional. Each can have a different decal.

14.2.2 MOD GATE

F2 MOD GATE is used to modify or create the Pin Decal for the

Connector Part. (Remember Pins are shown on connections, not Parts or Gates.)

When selected, the Reference Designation and Pin Text Block:



and the Origin appears. The P1-99 is a Block of Text that is reserved for the REF DES and Pin Number when brought up on one sheet of the schematic. It will read J1.2, J2.6, etc. Unlike PART TYPE, where the Decal is automatically created, you must create the Connector Decal Outline yourself. The menu changes to:

OTHER MENU
HIER/LIB
MOD GATE
1 2-D LINES
2 ADD TEXT
3
4 SETUP
5 COPY
6 MOVE
7 DELETE
8 ORIGIN
9 COMPLETE
0 EXIT

Except for the absence of TERMINALS, this menu is the same as a Gate Decal.

When F4 SETUP is selected, the following menu appears:

1 SELECT
2 PARAMETERS
3 MACROS
4 NAME DISP
0 EXIT

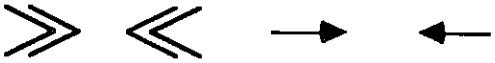
When F4 NAME DISPAY is selected, the SETUP CRT presentation is made with a special Red Pop-up Window as follows:

REF DES	ON
PACKAGE NAME	OFF
PART ATTRIB 1	OFF
PART ATTRIB 2	OFF

You may toggle these items On and Off with the cursor.

NOTE: The "Terminal Pad" for a Connector is the Decal ORIGIN. You will, therefore, after completing the Decal, want to position the Origin at the point on the Decal where the Connection will be made.

Generally, you will want to adapt some standard decal for your connections. These could take the traditional format of:



Because of the use of :

for Off-page symbols, you may not want to use this symbol for Connectors.

At the end of the modification, be sure to use F8 CHECK PART before saving the modified connector.

14.3 OFF PAGE

The objective of OFFPAGE is three-fold: (1) modify the Off-Page symbols, (2) modify Ground symbols, and (3) modify Power symbols.

Included in your software are Off-Page symbols, Power symbols and Ground symbols that have already been created for you. You may change these symbols using OFFPAGE, if you wish.

When selected, the menu changes as follows:

OTHER MENU
HIER/LIB
OFFPAGE
1 OFFPAGE
2 GROUND
3 POWER
0 EXIT

14.3.1 OFFPAGE

The OFFPAGE subroutine is used to modify the Off-Page Symbols used to indicate a connection is leaving one page and going to another, and in turn tie these two segments of the connection together.

To use, select F1 OFFPAGE and the Off-Page symbol will be brought up from memory and displayed at the CRT. This symbol has been created for your use by CAD Software, Inc. and has been stored with the name $OSR_SYMS.

In examining both the Electrical and Gate portion of the Off-Page symbol, you will notice that there are two symbols, one symbol indicating leaving the page and an alternate symbol indicating entering the page. This Primary symbol and its Alternate are Electrically the same and can be switched when being placed on the board by the use of ALTERNATE.

In examining the symbol itself, utilizing F2 MOD GATE, you will notice it contains three items: (1) the Outline of the symbol, (2) the Origin, and (3) the word "Net Name."

You are free to modify the Outlines of the Symbols using 2-D Lines if you choose to use an alternative way of indicating Off-Page.

The Origin is the point to which a connection is tied to the symbol (in lieu of a pin) and care should always be used in placing the origin such that when a connection is tied to the symbol, it is understood that the connection terminates at the origin location, wherever it may be with respect to the symbol.

The word "Net Name" is reserved for use in naming the Net which is going Off-Page. This Net Name is assigned by the Designer at the time the Off-Page Symbol is placed on the schematic. The options you have in changing Net Name are to change the size of the Net Name (as a subroutine of MOVE) or to relocate the location of the Net Name with respect to the Off-Page symbol.

14.3.2 GROUND

F2 GROUND is used to modify the Ground symbol provided with the software. The name of the Ground symbol is $GND_SYMS.

In examining the Electrical portion of the GROUND routine, you will note the Symbol Decal is named GND and the signal name that will always be associated with any connection which is connected to the Ground Symbol will be GND and, lastly, the Ground connection will always be 50 mils wide. (SMD Parts have Ground the same as the Pad Width, described in ***PADS-PCB***)

In examining the Ground symbol, utilizing F2 MOD GATE, you will notice that it consists of three parts: (1) the Ground symbol Outline, (2) the Origin, and (3) the Net Name. You may change the Symbol Outline if you wish, using 2-D Lines.

The Origin is the point of connection to which the connection to the Ground symbol is made. It can be moved, however, care should be taken to insure that insofar as the Schematic is concerned, connectivity is displayed between the connection and the Ground symbol.

The word Net Name is used to display the Net Name of GND. You may move it with respect to the Ground Symbol, change the size of the Text or suppress it from being displayed at the CRT or plotted on a sheet of the schematic.

Upon completion of the modification to the Ground Symbol, be sure to use CHECK PART before saving the modified symbol.

14.3.3 **POWER**

F3 POWER is used to modify the Power symbol contained in the software. The name of the Power symbol is $PWR_SYMS.

To use, select F3 POWER. In examining the Electrical portion, note that the Symbol Decal is +5V, the Signal Name is +5V and the Power track will be 50 mils wide. (SMD Parts have Power the same as the Pad Width, described in ***PADS-PCB***.)

The Symbol contains the three items of Component Outline, the Origin and the Reserved Net Name Space for the Power Net. These may be modified similar in fashion to modifications that could be made to the Ground Symbol and Power Symbol.

After modifying the Power Symbol, use CHECK PART before saving the modified symbol.

14.4 **REP OFFPAGE**

If you have changed the Off-Page, Power or Ground Symbol in the library, this routine allows you to replace the old symbol with the new symbol in the current schematic. To use, place the cursor over the symbol to be replaced (Offpage, Ground or Power), and press the left button. All occurrences of the selected symbol will be replaced with the new symbol from the library.

14.5 **GATE DECAL**

GATE DECAL is the routine used to create or modify the Decal used within the Gate when creating Parts. This routine is provided as a quick way of editing the graphical information in a decal without going through the Part Type editing routine. Although a liberal number of graphical Decals have been supplied with the ***PADS-LOGIC*** software, there are occasions when you may find you wish to create a new Decal or modify an existing Decal. F5 GATE DECAL is used. To use, select F5 GATE DECAL and respond to the prompt:

PART DECAL NAME (CR = TEST)>

with the name you want to use for the New Decal. In general, you should use the following criteria in naming your Decals. If the Decal is a unique Decal for a specific part, such as a Resistor, then name the Decal with this name. If, on the other hand, it is a generic part, such as a 2-Input Inverter, we suggest you use an abbreviation such as INV2, etc.

NOTE: Be aware that a Decal created with GATE DECAL is "Generic" in format; it has not been assigned to a Part Type, its Pin Numbers, etc. are not assigned at this time. All these finalization tasks are performed with the MOD GATE routine under the PART TYPE routine. If the Decal does not exist, you are advised and asked:

DECAL XXX DOES NOT EXIST -
DO YOU WISH TO CREATE DECAL (Y/N)?

With a Yes response, the CRT presentation is replaced with four lines of Text as follows:

1. Reference Designation Reserved Space — U999A
2. Package Name Reserved Space — PKG
3. Attribute 1 Reserved Space — ATTR1
4. Attribute 2 Reserved Space — ATTR2

The following prompt appears:

CREATE BOX SYMBOL AUTOMATICALLY (Y/N?)

If the Decal is to be a Rectangle or Square, then you should let the system automatically create the Decal Symbol. If, on the other hand, you want to make a custom Decal Symbol, then you would respond "N." The response of No would bring up a menu which will allow you to create the Decal with 2-D LINES, ADD TEXT, etc. A response of Yes brings up the prompt:

ENTER NUMBER OF INPUT AND OUTPUT PINS>

First, enter the number of Input pins you wish on the Decal, followed by a space and the number of Output pins. Following this, you will be invited to select the Pin Decal for the Input pin. Use the Pin Symbol Pop-Up Window that appears and use the cursor to select the Pin Symbols you wish for the Input pins. This will be followed by an invitation to select the Pin Decals for the Output Pins. Select the Output Pin Symbols from the Pin Symbol Pop-Up window.

Following this, the Automatically Created Box Decal, together with the Input and Output pins, and Reserved Text for Reference Designation, Part Type and two Attributes are displayed at the CRT. The menu is changed to allow you to use 2-D LINES, ADD TEXT, ADD TERMINAL, select the Text to be displayed, etc.

Rearrange the Reserved Text Boxes, relative to the Symbol as you desire. The REF DES is usually placed inside an IC Gate, and above a Discrete Part. The PKG Attribute Lines are usually located below the Decal.

Upon completion of the creation of the Gate Decal, select F9 COMPLETE and confirm that you wish to store the Decal in the User's Library, under the name originally selected. The Gate Decal is saved in the library utilizing this new name.

The routine of GATE DECAL is also used to modify a Decal that you have created or to modify a Decal that has been provided for your usage by CAD Software, Inc., so as to create the New Decal.

To use, select F5 GATE DECAL and respond to the prompt:

PART DECAL NAME (CR=TEST)>

with the name of the Decal you wish to modify.

NOTE: To obtain a complete listing of all the Decals that are in the software utilize, the wild card * and a Red Pop-up Window will appear, listing all of the Decals that are stored. Select the Decal to be modified. The Decal will be displayed at the CRT, together with the menu that permits all features of the Decal to be modified.

Upon completion of the modification, use F9 COMPLETE, then SAVE.

14.6 PIN DECAL

F6 PIN DECAL is used to create or modify the Pin graphics that are associated with the Gate Decals. It will be remembered that when working with Pins, while creating a Gate Decal, a Red pop-up Pin Symbol Window lists several different types of Component Pins. These different types of Pins have seven different symbols and in general can be used to indicate various types of data, such as Clock lines, Inputs, Output, etc. The Pin decals are Graphic symbols and do not have logical significance.

When F6 PIN DECAL is selected, the menu changes to:

OTHER MENU
HIER/LIB
1 SELECT
2 NEW PIN

and the Red Pop-up Window with a listing of Pins appears. If you wish to modify one of these Pin Decals, use F1 SELECT and select with the cursor. Make the modification and Save.

If you wish to create a new Pin Decal use, F2 NEW PIN and respond to:

ENTER PIN DECAL NAME TO CREATE (CR=TEST)>

with the name of the New Pin Decal you wish to create.

Four items of Text will appear at the CRT. These are:

#E
PNAME
#0:TYP=U SWP=0.
NET NAME

These Text blocks are reserved spaces for various Attributes that you can assign to the Pin as follows:

#E - The location reserved for the Pin Number to be added when the Gate is created.

PNAME - This defines the location of the Pin Name relative to the pin graphics.

#0: is used to indicate the sequence in which Pins were added to a Part.

TYP=U - refers to the "Logical" type of the Pin and is used during the "Checking" process, run upon completion of the capture of the design. The type choices are:

S = Source Pin
B = Bidirectional Pin
C = Open Collector Pin
O = Or-Tieable Source Pin
T = Tri-State Pin
L = Load Pin
Z = Terminator Pin
P = Power Pin
G = Ground Pin
U = Undefined

SWP=0 - This indicates the swappability of the pin with other pins. 0 indicates the pin cannot be swapped with other pins; a 1 or Other Number indicates that it is swappable with other pins on the same gate with the same Swap Number.

NET NAME - This is the Text location reserved for the Net Names as assigned to the connection going to the pin, either by the system or the designer.

NOTE: You do NOT change or set up any of the Pin Text while creating a New Pin. This task is done when creating a Gate for the Part, at which time information for finalizing the Text will be available. You should, however, place these names as precisely as possible, using the correct text justification to minimize the amount of pin text placement when creating the gate decals.

The menu changes to the following:

OTHER MENU
HIER/LIB
1 2-D LINES
2 ADD TEXT
3
4 SETUP
5 COPY
6 MOVE
7 DELETE
8 ORIGIN
9 COMPLETE
0 EXIT

In proceeding to make up a new pin, proceed as follows:

First, use 2-D Lines to create the pin symbol.

Next, locate the origin at the point on the symbol you wish the connection to attach to the pin. This is done by placing the cursor at the point you wish the origin to be and selecting F8 ORIGIN.

Next, using MOVE, locate the four Reserved Boxes with respect to the Pin Symbol in the fashion you wish them to be on the Schematic.

Finally, turn ON/OFF each of the four Reserved Text Boxes as appropriate, using NAME DISP under the SETUP menu. Complete the New Pin using F9 COMPLETE. The New Pin will be added to the Red Pin Symbol Pop-up Window.

14.7 THE PKGTYP.DAT FILE

Within the ***PADS-LOGIC*** directory is a file called PKGTYP.DAT. This file is used to control the Reference Designation stem of a Part when brought on to a sheet of Schematic.

The file and supporting features perform as follows:

1. When creating a New Part, the ELECTRICAL section has a data

line called LOGIC FAMILY. Enter in the Data Line the Logic Family of the New Part, i.e., TTL, CMOS, RES, CAP, CON, etc.

2. Ensure the PKGTYP.DAT file has the Logic Family contained in the file, together with the appropriate REF DES stem, i.e., RES = R.

3. When a Part is brought onto the board, the Stem of the REF DES is obtained from the PKGTYP.DAT file and the Sequence Number, i.e., R1, R2, etc., is obtained from the Accounting module.

4. If, in the creation of a New Part, the Logic Type is left undefined, when this part is brought onto the schematic, a prompt asks for the REF DES Prefix.

To print a copy of PKGTYP.DAT, Exit to the PADS Directory and in response to: C:\PADS>
type: PRINT PKGTYP.DAT <CR>

The following is an example of the PKGTYP.DAT file:

```
RES  R
TTL  U
CMO U
ECL  U
ANA U
CAP  C
DIO  D
ZEN  Z
TRX  Q
XFR  T
CHO L
CON J
RLY  RL
SCR  TH
TRC  TH
SWI  SW
```

Use PEDIT to change the file.

Chapter 15

HIERARCHICAL ROUTINE

15.0 HIERARCHICAL BACKGROUND

The Hierarchical Routine is a Graphical short hand technique used to display or plot a maximum amount of data with a minimum of graphical information. Hierarchy is especially useful in those situations wherein the same circuit is used repeatedly within the design. Example: A specific circuit described by a sheet of Logic is used fifty different times in your design. By making this circuit a Hierarchical Symbol, you need plot the sheet only one time rather than 50 times, yet the complete Parts and connection data for all fifty usages are in the database of the design.

Hierarchy is also useful to show a higher level "block diagram" representation of the schematic.

The concept of "Levels of data" is used in Hierarchy. Associated with Levels are the concepts of Top-Down (from the Top level down to the Bottom level) and Bottom-Up, as well as a "Flattened design."

Perhaps the best analogy for illustrating the concept of Hierarchy is to compare it to something with which we all are familiar. An excellent (although not exact) analogy to Hierarchy is the various levels of Political units (USA, States, Counties, Cities/Towns) we have within the United States, combined with our Road or Highway system.

Consider a map of the United States that shows all fifty states with the States represented by symbols. In this case, the symbols are depicted in the shape of the State. All States are at the same level, that is to say, the Top Level or Level 0 on the USA map. Similarly, in the ***PADS-LOGIC*** Hierarchical concept, we have a Top Level Drawing or Drawings. This Top-Level Drawing(s) can be likened to the map of the United States in that the Top Level drawing has symbols depicting the next lower Hierarchical level within a design. Note that whereas on the United States map, the State symbols are

"shaped as per the State," in the ***PADS-LOGIC*** Top Level drawing, the symbols are rectangle boxes with the item represented by the Rectangle Box symbol. In both instances, States or Hierarchical Symbols, they are immediately below the Top level and are considered to be at Level 1.

The next level in the Government Hierarchy below the United States is a State. Similarly, the next level below the Top Level (Level 0) in the ***PADS-LOGIC*** Hierarchy is a Hierarchical symbol containing circuitry (and perhaps another Hierarchical Symbol for the next lower level) part of the design. Usually, but not necessarily, this level would contain a specific circuit such as a counter, latch, power supply, etc., consisting of a number of parts connected together to provide this circuit function. It is possible to divide the gates of multigated IC's on different Hierarchical levels, for example. The ***PADS- LOGIC*** system takes care of this automatically for you. This Top Level Drawing(s) consists of the Numbered Sheets, i.e., Sheet 1, Sheet 2, etc., and it may consist of circuitry and Hierarchical symbols or just Hierarchical symbols only.

NOTE: It is important to appreciate that in our concept of a Hierarchical Design, Numbered Sheets, say Sheet 1 through 6 (as long as they have sheet numbers) are all considered as part of the Top Level, unless they represent the contents of a Hierarchical Symbol, in which case, the sheet is at whatever level the Hierarchical Symbol has been defined and is NOT included in the Numbered Sheets of the Design. Think of it this way: If you could paste these sheets together and include all the Hierarchical Sheets as well, the "pasted results" would all be at the same level and would contain the entire Design. We should consider this Top level represented by all the Sheets as Level 0 or a "Flattened" Design.

NOTE: A "Hierarchical Sheet" as opposed to a "Numbered Sheet" will not be included in the Sheet Numbering sequence of the design.

At this point, we deviate somewhat from our analogy with the United States map. In the instance of the United States map, we would never have multiple symbols for the same State on the United States map since a state only occurs once. With the Top Level drawing, however, we could have the Next Lower Level Numbered symbol repeated any number of times, indicating that this particular

circuit is used a number of times at the next level. Incidentally, the Auto REF DES Assignment System will correctly assign REF DES, even when the same circuit is used N times.

We have seen in our Map Hierarchical presentation, the first Level below the Top Level (United States) is the individual State; Level Two would be the Counties within the States and the Level Three would be the Towns within the Counties. So it is with our ***PADS-LOGIC*** Hierarchical Level.

Again, we come to a diversion between our Governmental analogy. For example, in Massachusetts, there is a County (Level Two) called Middlesex County (where the American Revolution started). In our Political Governmental structure, we could NOT have a Middlesex County (identical to the Massachusetts Middlesex County) anywhere else in the United States. In the ***PADS-LOGIC*** Hierarchical system, however, this is not the case. One could have a latch circuit, for example, duplicated anywhere within the design and at any level. We simply create the latch circuit by assigning to it a Hierarchical Symbol, create a Hierarchical Sheet containing the latch circuitry, then place the Symbol at any level within the Hierarchical structure.

A Hierarchical Symbol placed on any of the top level sheets, would indicate that symbol and its contents would be at Level 1. If the Hierarchical Sheet represented by the Hierarchical Symbol contained circuitry and another Hierarchical Symbol, then this Hierarchical Symbol and its contents would therefore be at Level 2, etc.

And now to our analogy between the Highway System within the United States and ***PADS-LOGIC*** Hierarchy. The famous (but no longer in existance) Route 66, that formerly traversed the United States from the East Coast to the West Coast, crossed many States (Level 1)) went trough many Counties (Level 2), and many Cities, Towns and Villages (Level 3). Similarly, a connection with the same name that is in a design captured by the ***PADS-LOGIC*** Hierarchical system can traverse all Hierarchical levels and the Top Level. In the process, it is tied to Nodes (part pins) that correspond to Cities, Towns and Villages in our Government Hierarchy. This connection net is identified in our Hierarchical design through the use of same Net Name. A listing of all the Cities, Towns and

Villages through which Route 66 traveled would be similar to a listing of all the Nodes associated with a particular Signal Net (Net List).

Our final analogy: The concept of "Flattened Design." If we go back to our Government Map analogy, the concept of a very large scale United States, large enough to show not only the States, but the Counties and the Cities, Villages and Towns, this would be a "flattened" Governmental Hierarchical presentation. Our Top Level sheets in our drawing structure capturing the Design plus all the Hierarchical Sheets represents the concept of a flattened design in that the contents of these Top Level sheets and the Hierarchical sheets contain within the database of all of the sublevels depicted. When you output all the sheets, the ***PADS-LOGIC*** system flattens the contents for you and outputs all the parts, connections, etc. in the entire design including Numbered Sheets and Hierarchical Sheets.

The concept of flattening is not limited to the Top Level. You can take a section of a Design including some Numbered Sheets and some Hierarchical, and create a Second Design with these sub parts (use GROUP), create a new file of the second design and output it as a Flattened subdesign.

15.1 NEW HIERARCHICAL COMPONENT

The Hierarchical Routine is found under the HIER/LIB routine, as F7 HIERARCHY. When selected, the menu changes to the following:

```
OTHER MENU
HIER/LIB
HIERARCHY
1 NEW HCOMP
2 MOD HCOMP
3 ADD SHT NO
4 DEL SHT NO
8 HIER IN
9 HIER OUT
0 EXIT
```

The routine New Hierarchical Component (F1 NEW HCOMP) is used to create the symbol for a New Hierarchical Component or Hierarchical Symbol. When F1 NEW HCOMP is selected, the prompt

ENTER HIERARCHICAL SYMBOL NAME>

is displayed (see end of Section 12.1 for an answer of No to the prompt). Respond to this prompt with the name that you wish the symbol to be called.

You will then be presented with the prompt:

CREATE NEW HIERARCHICAL SHEET (Y/N?)

If a response of Y is selected, the new Hierarchical component will be associated with a blank unnumbered Hierarchical sheet that will be created automatically. A response of "Y" brings up the CRT presentation used to create a Hierarchical Symbol. This presentation consists of the Name of the Hierarchical Symbol, together with the Origin Symbol and the prompt:

ENTER NUMBER OF INPUT AND OUTPUT PINS>

is displayed. You respond by first entering the number of Input pins that you wish on the Symbol, a space, followed by the number of Output pins.

The following prompt is displayed:

SELECT PIN DECAL FOR INPUT PINS

A Red Pin Symbol pop-up window will appear with a listing of different types of pins as follows:

PIN SYMBOLS
PIN
PINB
PINCLK
PINCLKB
PINIEB

PINORB
PINSHORT
PINVRTS

These different pins represent different pin symbols that have been prepared for your use. Using the cursor, select the pin whose symbol you want to use for the Input pins, then in response to the prompt:

SELECT PIN DECAL FOR OUTPUT PINS

select the pin symbol that most nearly corresponds to the symbol you wish to use on the Hierarchical Symbol for the Output pins. When this is done, a Yellow square or rectangle will be created at the CRT. The square or rectangle represents the Hierarchical Symbol Outline and will contain the Hierarchical Symbol Name (at top), the Symbol Origin (at lower left corner), and the chosen Input Pin Symbols on the left hand side and the chosen Output Pin Symbols on the right side.

The menu will change to as follows:

OTHER MENU
OTEHR MENU
HIERARCHY
1 2-D LINES
2 ADD TEXT
3 TERMINALS
4
5 COPY
6 MOVE
7 DELETE
8 ORIGIN
9 COMPLETE
0 EXIT

With this menu, you are able to finalize the Hierarchical symbol.

2-D LINES

The selection of F1 2-D LINES brings up the 2-D LINE menu and

you may manipulate the Outline of the symbol. (It is a 2-D Line.) You may manipulate it in the same fashion as 2-D Lines, to include saving the symbol outline and recalling it. Bear in mind that the manipulation you use on the symbol with the 2-D LINE routine is merely to change the outline of the symbol. For example, if you want to use COPY and create another symbol, this simply results in a Hierarchical symbol depicted by two Outline symbols. This does not, REPEAT, does NOT mean that you will have two Hierarchical circuits. There is only one circuit associated with each named Hierarchical symbol, irrespective of how "fancy" you make the circuit symbol Outlines.

You may even save a "highly creative" symbol outline if you wish using F9 TO LIBRARY, bearing in mind, however, that all you're doing is saving the Symbol outline, not its circuit content.

ADD TEXT

The subroutine F2 ADD TEXT allows you to add free text to the circuit symbol. This free text may be combined with the Hierarchical Symbol Outline by using the COMBINE feature of 2-D LINE.

TERMINALS

F3 TERMINALS can be used to add to or change the Terminals that have already been added to the symbol. When F3 TERMINALS is selected, the menu changes to the following:

OTHER MENU
OTHER MENU
HIERARCHY
TERMINALS
1 ADD TERM
2 CNG DECAL
3
4
5 SET PNAME
6
7
8

9 CNG PNAME
0 EXIT

ADD TERMINAL is used to add additional Terminals above and beyond the Input and Output Terminals, defined at the time the symbol was automatically created. To use, select F1 ADD TERM and the Pin Symbol pop-up window appears. Select the appropriate pin with the cursor and rotate or mirror it to bring it to the correct location on the Hierarchical Symbol. If you find you have selected the wrong pin decal, use F2 CNG DECAL to change the Decal. When in the correct location, finish with F1 COMPLETE.

NOTE: A Pin Name on a Hierarchical component specifies the Net Name of any connection that is tied to that pin of the Hierarchical component. All pins on a Hierarchical component must have a Pin Name.

CHANGE DECAL

To change a Terminal Decal that was created when the Hierarchical Symbol was originally created or was added, using ADD TERMINAL, select F2 CNG DECAL and then select the new Pin Terminal from the Pin Symbol pop-up window and, when selected, then select the Pin Symbol to be changed with the cursor. The current Pin Symbol will be replaced with the selected new Pin Symbol.

SET PIN NAME

Each pin on the Hierarchical Symbol must have a Name which is used to tie or "connect" that particular pin in with other similar pins on other Hierarchical Symbols or the Top Level Drawing, so as to create a complete connection net across all levels. Use F5 SET PNAME to assign the name to the pin. To use, place the cursor over the Pin Symbol to be named and select F5 SET PNAME. In response to the prompt:

ENTER PIN NAME (CR = NONE)>

enter the desired name of the pin. This Name can be either an Alpha or Numeric character or a combination of Alphanumeric characters.

Place the cursor over the pin to be named and the assigned pin name will appear. An Automatic incrementing feature can be enacted through the use of the left mouse button, which will increment the pin name if an Alpha character is used (from A through Z). If Numerics are used, the Automatic incrementing will increment from 0 through 999. If an Alphanumeric name is used, for example 1A, the Automatic incrementing will increment from 1A through 1Z.

CHANGE PIN NAME

F9 CNG PNAME is used to change the name of a pin previously named. To use, place the cursor over the name of the pin to be changed and respond to the prompt:

CHANGING PIN NAME ZZ -
ENTER NEW PIN NAME>

with the new pin name. The Pin Name will be changed from the old pin name to the new pin name.

NOTE: Pay particular attention to the assignment of pin names inasmuch as this is the intelligence used to tie together a Signal Net that may go between various Hierarchical levels to and even within the top level drawings of the design. All instances of the Signal, regardless of where found in the design, using a common pin name or signal name, will be tied together into a single, contiguous Connection net and treated as such in the Net List.

COPY

COPY is used to copy Pins or the Outline only, of the Hierarchical symbol. To use, place the cursor over the object to be copied and select F5 COPY.

MOVE

MOVE is used to move either a Pin or the Hierarchical Symbol Outline. When the Outline is moved, the remaining data, i.e., Pins, Text, etc. does not follow.

DELETE

DELETE is used to delete the Hierarchical Symbol Outline. Additionally, DELETE can be used on an individual basis to delete pins that have been added to the Hierarchical Symbol, providing the Hierarchical Symbol Outline is present.

ORIGIN

F8 ORIGIN is used to relocate the origin from the location in the lower left hand corner of the circuit symbol, wherein the origin was automatically placed when originally created. To use, place the cursor at the new desired location of the Origin and select F8 ORIGIN. The Origin will be relocated to the cursor location.

COMPLETE

F9 COMPLETE is always, repeat always, used to complete the creation of the Hierarchical Symbol. Use F9 COMPLETE when you have placed all the pins desired on the symbol, any 2-D Lines you wish, any Text you wish and you have named the pins.

NOTE: The Hierarchical Symbol cannot be completed if all of the pins have not been defined with Pin Names.

When F9 COMPLETE is selected, the system goes "WORKING" for a few seconds and the newly created symbol shows up on the CRT in the White color and is attached to the cursor. Move the cursor to the location wherein you wish the newly created symbol to be located, and select F1 COMPLETE.

In the sequence which you have observed above, you have created a Hierarchical Symbol whose circuit will be Automatically contained on a sheet that is NOT defined by the regular Sheet Number sequence used in the design, but rather is defined as a Hierarchical Sheet.

The preceding discussion has been made under the assumption that you respond Yes to the above prompt.

A RESPONSE OF NO TO "CREATE NEW HIERARCHICAL SHEET (YES/NO)"

The decision for responding Yes or No is based upon the following: If you wish the contents of the Hierarchical Symbol to Not be contained on one of the numbered sheets in the drawing, but rather on a separate entity called a Hierarchical Sheet, you would respond YES. On the other hand, if you don't care for this (and the choice is certainly yours), you can have the symbol represent the contents of an existing numbered sheet of the schematic. When a Hierarchical Symbol represents an existing numbered sheet, a pin is created for each off-page symbol on the sheet and the pin name will have the net name assigned to the off-page symbol. In either instance, the decision is yours. The system functions in the same fashion in terms of collecting the data for both Hierarchical and Numbered Sheet drawings and flattens them out for the Net List, independently of the type of sheet they are on.

A response of NO will bring up the prompt:

ENTER SHEET NUMBER FOR HIERARCHICAL SYMBOL>

This prompt is asking you what sheet in the Numbered Drawing Sheet sequence you wish the contents of this symbol that you are creating to appear upon. Let's say you had defined a 3-sheet Drawing and you wanted the contents of this Hierarchical Symbol you are creating to show up on Sheet #3, then you would respond to the prompt with 3.

The CRT presentation will show the Hierarchical Symbol Name and the Sheet Number and the Origin, together with the Red pop-up window for pin symbols and you will be asked in the prompt to select Pin Decal for Input Pins. Using the cursor, select the Pin Decal you wish for Input Pins.

You will then be asked to select the Pin Decal for Output pins. Again, using the cursor, select the symbols for the Output Pin. The Hierarchical Symbol will then appear with the Name on top and the Sheet Number on the bottom, with the Origin in the lower left hand corner.

The number of Input and Output pins is determined by the current contents of the numbered sheet that the Hierarchical Symbol represents. If the sheet has no Off-page symbols, no pins are created automatically. You may add pins using the ADD TERM routine.

When finished, ALWAYS use F9 COMPLETE. The sheet on which you are creating the Hierarchical Symbol will contain the symbol you have created in White, attached to the cursor. Move the symbol to the location desired and place, using F1 COMPLETE.

15.2 MODIFY HIERARCHICAL COMPONENT

F2 MOD HCOMP is used to Modify the Hierarchical Symbol, representing the Hierarchical level. It is not used to modify the circuit content of the Hierarchical Symbol. To use MOD HCOMP, place the cursor over the Hierarchical Symbol to be modified, and select F2 MOD HCOMP. The selected Hierarchical Symbol will appear at the CRT and the menu will change to the following:

OTHER MENU
HIER/LIB
HIERARCHY
MOD HCOMP
1 2-D LINES
2 ADD TEXT
3 TERMINALS
4
5 COPY
6 MOVE
7 DELETE
8 ORIGIN
9 COMPLETE
0 EXIT

You will note that this menu is the same as was available in the construction of the original Hierarchical Symbol and allows you to Add 2-D Lines, Text, Terminals, as well as to Copy, Move and Delete the Symbol Outline and, finally, relocate the Origin. Upon completion of the modification of the Hierarchical Symbol, use F9 COMPLETE.

15.3 ADD SHEET NUMBER

If you wish to assign a Sheet Number to a Hierarchical Symbol and have the Sheet Number appear as Text on the Symbol, as well as act as a "Pointer" to the Sheet, use F3 ADD SHT NO. In response to:

SELECT HIERARCHICAL COMPONENT>

select the Hierarchical Component with the cursor. If the Hierarchical Symbol already has a Sheet Number, it cannot be selected. If it does not have a Sheet Number already assigned, the system will assign it the next highest available sheet number.

NOTE: Reminder - This Sheet Number is a Number within the Sheet Number sequence of the design, NOT a Hierarchical Sheet Number.

15.4 CREATING THE CIRCUIT CONTENTS OF A HIERARCHICAL SYMBOL

To this point in the discussion on Hierarchy, we have concentrated on the creating of the Hierarchical Symbol. We have not addressed how one creates the circuit contents of the Symbol and moves between the Hierarchical Symbol and its circuit. This is done in one of two ways: (1) Using the HIERARCHY IN/HIERARCHY OUT subroutine, or (2) the Modeless Commands HI or HO.

15.4.1 HIERARCHY IN

The subroutine F8 HIER IN allows one to look inside or push down inside a Hierarchical Symbol. To use, place the cursor over the Hierarchical Symbol that you want to look inside and select HIER IN. Alternatively, place the cursor over the Hierarchical Symbol and use the Modeless Command HI. In either instance, a sheet representation appears at the CRT that contains the circuit contents of the symbol if it has already been created. If it has not already been created at this point, then use an Exit back to the routines that allows you to add components, add connections, buses, text, 2-D Lines, etc. and construct the circuit. During the construction of the circuit, pay particular attention to the Input and Output pins, to ensure that the connections within the Hierarchical Symbol will be

connected to the appropriate connections outside the Hierarchical Symbol.

Once finished, go back to the Hierarchical Routine and use F9 HIER OUT to move back up to the Hierarchical Symbol representing the circuit just completed.

15.4.2 HIER OUT

F9 HIER OUT or Hierarchy Out allows one to move Up in the levels of the Hierarchy. For example, if one had the circuit contents of a Hierarchical Symbol displayed, and selected F9 HIER OUT, the circuit contents would be replaced with the Hierarchical Symbol.

The Modeless command HO performs the same function.

NOTE: In some Hierarchical schemes, the concept of Hierarchical In and Hierarchical Out can be compared to "Push" and "Pop," where "Push" allows you to push down inside levels as does Hierarchy In and "Pop" allows you to push up from the lower level to a higher level.

15.5 NOTES ON HIERARCHICAL LEVELS

It is possible to become confused as to what level a Hierarchical Symbol is on. Let us, therefore, review the concept of Levels.

Level 0 should be used as the Top Level or Level 0, containing the contents of the design that are represented by Numbered Drawing Sheets. This can be any number from 1 sheet up to 100 sheets. As a recommendation, we suggest that if indeed you plan on having a number of Hierarchical levels in the design, that you utilize Sheet #1 to display all or most all the Level One Hierarchical Symbols. Sheet #1, however, can be a mixture of the design circuit as well as all or part of the Level 1 Hierarchical Symbols.

The name you assign to a Hierarchical Symbol should give a clue as to what level the symbol occupies in the level order. For example, the last character in the name of all of the Level 1 Hierarchical Symbols could be a 1. Example: AAA1, BBB1, CCC1, etc., Level 2 Symbols could be ZZZ2, YYY2, etc. In this fashion, you will

immediately be aware of the level.

There is no limit within ***PADS-LOGIC*** to the Hierarchical levels that you can create.

Moving Up and Down from a level is quite easy using the F8 HIER IN and F9 HIER OUT subroutine.

The Modeless commands of HI and HO can be used in any menu.

Note that the Purple window in the upper left hand corner contains information on the sheet or Hierarchical Symbol currently displayed. This is contained in two lines of data. The first line of data is the name of the schematic. The second line of data is the current sheet being displayed, whether it be a sheet contained within a Hierarchical Symbol or a sheet within the numbered sequence of the design; or if it is a Hierarchical Sheet, the name of the Hierarchical Symbol.

15.6 FLATTENED DESIGN

Creating a Net List of the Design Automatically Flattens the design and creates a listing of all Parts, with the correct Automatically assigned REF DES and Connections. Simply have the Design in memory and select IN/OUT, REPORTS, and NET LIST (or other reports you may wish).

15.7 SUB-SECTIONS OF A HIERARCHICAL DESIGN

In certain instances, such as Logic Simulation, you may NOT desire the complete design, but instead wish only a portion of the design, for use in creating the Net List to be simulated. Proceed as follows:

1. Using GROUP, cut out and save those sheets including the Hierarchical Symbols and their circuits as desired.

2. Start a New Design and paste the groups into this New Design.

3. Output the New Design as per Section 12.6

Chapter 16

THE MACRO ROUTINE

16.1 WHAT IS A MACRO?

A Macro is a collection of keyboard and mouse events that can be replayed immediately (Auto Macro) or at a later time (User Macro), and in the process, save a considerable amount of the Designer's time, since one Macro can contain many, many individual commands. In short, Macros are a form of Command Shorthand.

A Macro is replayed and performs its intended function when it's name is typed. A macro can be named using any one of the following methods:

1. ALT A through ALT Z (case insensitive)
2. ALT F1 through F10
3. SHIFT F1 through F10
4. CONTROL F1 through F10

All macros are stored in memory for quick replay. The number of macros that can be stored is limited by the number of names available (currently 56) and the amount of memory available (actual limits are discussed later). The total number of keystrokes for all macros that can be stored is limited by available memory. There is no limit on the number of keystrokes for an individual macro other than the memory limit.

The Macro Management Menu is found in SETUP, F3 MACROS. When selected, the menu changes to the following:

SETUP
MACROS
1 HELP/STAT
2 LOAD FILE
3 SAVE
4 DEL MACRO
0 EXIT

16.2 CREATING A MACRO

Macros are created one of two ways: from within ***PADS-LOGIC*** "By Example" or by using a Text Editor and writing a macro file that will be loaded later at run time. Both forms of creation and the format of a macro file are discussed later.

16.3 DISPLAY RESOLUTION INDEPENDENCE AND COORDINATES WITHIN MACROS

Whenever anything on the screen is picked during macro creation, whether it is a component or a choice in a pop-up menu, a coordinate is stored in the macro. If that macro is played back with ***PADS-LOGIC*** configured for a graphics adapter with a different resolution, the stored coordinates must point to where the user originally intended. This is referred to as resolution independence. Even though the above situation probably won't happen that often, it's good to know a little something about it. Picking objects on the schematic, i.e., components, connections, etc., doesn't pose much of a problem for resolution independence as those "database" coordinates are by definition, already independent of display resolution. The ones to worry about are picks in pop-up menus and forms. The macro system tries to store these coordinates in as resolution independent a fashion as possible. The user can help by picking these objects as close to the center of the object as possible.

16.4 ABSOLUTE VERSUS RELATIVE COORDINATES

Macros can have two kinds of coordinates: Absolute and Relative. An Absolute coordinate is independent of all other coordinates in that macro. A relative coordiante is always related to its immediate predecessor (i.e., move 100 units in the X and -50 units in the Y from the previous coordinate.)

The User Macro always defaults to the "ABSOLUTE" coordinate mode. A macro with Absolute coordinates, when replayed, always generates the same coordinates with which it was created. A good example is a macro that fills a form or picks a choice from a pop-up. Forms and pop-ups always appear at the same place on the screen and, therefore, the macro should always replay at that place.

A macro with relative coordinates, when replayed, generates coordinates that are relative to the cursor position at the time replay starts. An example is a macro that does some repetitive, step and repeat function, such as connecting a memory to an address bus, or picking an array of parts of the schematic. The user would create a macro that connects one line (using relative coordinates). The cursor is then moved to the next pin to be connected and the macro is replayed. Since all the coordinates in this macro are relative to the cursor, this pin will be connected in the same fashion as the first.

16.5 THE AUTO MACRO

There is a special kind of macro called the Auto Macro. It does not need a name and hence there is only one at any given time. (The other type of macros, the ones with names, are called "User" macros.) The Auto macro exists mainly as a quick way to make a temporary macro that you intend to use only once and then discard. This macro cannot be saved to a file.

16.6 LOADING A MACRO FILE

Macro files can be loaded two ways. One way is to put the macros that you always want loaded in the file CAE.MAC in the LOGFIL directory. This file is always automatically loaded at program startup. The second way is to load from within ***PADS-LOGIC*** by using the LOAD Command on the MACRO menu. The LOAD command will always load all the macros in a file. If any of the macros in the file already exist in memory, the load process will be aborted.

16.7 MACRO REPLAY CONTROL

There are two types of control you have while a macro is replaying: Pausing and Aborting. To pause during replay, use the Space bar. To resume, use the Space bar again. To abort replay, type ^X (Control-X).

16.8 YES/NO QUESTIONS

There are many places in the software where a Yes/No question needs to be answered while creating a Macro. These include various

phases of Part Editing as well as when Exiting the system. Not all of these questions are asked every time. For instance, if you try to save over an existing part or file, you will be asked to verify. When creating a macro you need a way to deal with this situation.

The way to control this is with the Yes/No mode. This mode tells the macro system how to deal with Yes/No questions. The Yes/No mode affects both the way macros are created and the way they are replayed.

For instance, if the mode is anything other than M (Macro) and a Yes/No question is encountered during macro creation, the answer to that Yes/No question is not stored in the macro. During replay, if the mode is anything other than A (Ask), the question is answered either by the system or from the macro. There are several modes--each is discussed below.

mode = Y (yes)	During replay, always answer YES. During creation, don't store the answer to any Yes/No quesitons.
mode = N (No)	During replay, always answer NO. During creation, don't store the answer to any Yes/No questions.
mode = A (Ask)	During replay, if a Yes/No question occurs, the user is asked for the answer. During creation, don't store the answer to any Yes/No questions.
mode = M (Macro)	During replay, if a Yes/No question occurs, get the answer from the macro. During creation, STORE the answer to any Yes/No questions.
mode = R (Restore)	(see below)

16.9 YES/NO RESTORE MODE

This mode provides a way for a macro to temporarily change the Yes/No mode and then restore it to what it was previously. If a

(Y/N) command is encountered during replay, the old mode is remembered by the macro system. When the (Y/N) command (Set Yes/No mode, restore) is encountered, the mode is restored.

16.10 MACROS CAN CALL OTHER MACROS

If a macro is invoked during macro creation, the call to that macro is stored in the one being created.

16.11 MACRO COMMANDS

The following is a summary of the macro related commands. Following the summary, some of the commands are discussed in detail.

* alt 1 = replay auto macro
* alt 2 = create/save auto macro
* alt 3 = create/save user macro
* alt 4 = set absolute/relative coordinate mode
* alt 6 = set yes/no mode
* alt 7 = restore menu position
* alt 8 = go to top menu
* alt 9 = display file
* alt 0 = display errors
* C = set macro replay count
* F1 = Help/Status
* F2 = Load file
* F3 = Save macros to file
* F4 = Delete a macro

16.12 CREATE AUTO/USER MACRO

These commands are toggles, hit it once to begin creating a macro, hit it again when the macro is complete. When creating a user macro, you will be asked for the macro name (the key used to invoke replay of the macro) and an optional description. When user macro creation is complete, you will receive a message to that effect, including the current status of the macro system. The status tells you how many macros are defined, their total length (in keystrokes) and the percentage of available macro memory in use.

When you complete creation of a macro, the current cursor position is stored in the macro as a MOVE command.

16.13 SET ABSOLUTE/RELATIVE COORDINATE MODE

This command toggles the mode between Absolute and Relative. As stated above, CREATE AUTO MACRO always starts in Relative mode. CREATE USER MACRO defaults to the Absolute mode.

The following is a VERY IMPORTANT NOTE: When creating a User Macro, using Relative coordinates, make sure the coordinate mode is toggled from Absolute to Relative before the first coordinate is stored. This initializes the relative coordinate system and ensures that the first relative coordinate is relative to the cursor.

16.14 RESTORE MENU POSITION

Whenever a macro is invoked, the current menu position is stored. The restore command, when encounted during playback, will attempt to place the user at that saved position. When encountered during creation, this command will write a restore command to the macro (it will not do a restore).

There are certain situations in which the restore command will not restore to completion. These include inconsistent Yes/No questions and menu commands that require the user to type a response (such as entering a Decal Name to MOD DECAL).

16.15 TOP MENU

This command will attempt to move to the top most menu. The operation of this command is tied closely to that of the Yes/No command since many Yes/No questions may be encountered on the way to the top. If this command is encountered during macro creation, a Top command is written to the macro and then a Top command is performed.

16.16 DISPLAY FILE

While this command is not part of the macro system, this is where

you'll find out about it. This command displays a file in the pop-up window. Lines that are too long are wrapped around. Files that are too big are truncated. If no path is given in the filename, the LOGFIL directory is assumed. While the file is displayed, the following commands are available:

Home	Display first screen
End	Display last screen
PgUp	Display previous screen
PgDn	Display next screen
Esc	Exit file display

16.17 DISPLAY ERRORS

This is a special version of the Display File command. If error messages are being redirected to file CAE.ERR, this command will display that file. If errors are going to the printer, you'll receive a message to that effect.

16.18 SET MACRO REPLAY COUNT

This command specifies the number of times a macro will be replayed when it is invoked. It is primarily for use with auto macros but can be used with user macros. To replay a macro only once, just type the macro name. To replay it more than once, type "c" followed by the number of times you want it to replay (no <CR>), then type the macro name. For instance, to replay the alt-a macro 3 times, type c3 and then alt a.

This command only affects the current macro replay. The next macro to replay (in the absence of a "c" command) will replay only once.

The "c" command can be stored in a macro. Therefore, a macro A can call macro B 3 times.

16.19 HELP/STATUS

This command displays in a pop-up window (using the pop-up window commands from above) some macro help text as well as the current status of the macro system. This status includes one line for

each defined macro, each line containing the following:

* the macro name
* the length of the macro in keystrokes
* the percentage of macro memory the macro occupies
* the macro discription, if any

The last line gives the total-length of all macros and the total percentage of macro memory used.

16.20 LOAD FILE

This command is used to load a macro file to memory. If no path is given in the filename, the LOGFIL directory is assumed.

16.21 SAVE FILE

This command saves (writes) ALL the currently defined macros to the specified file. If no path is given in the filename, the LOGFIL directory is assumed. This command will OVERWRITE any existing file of the same name.

16.22 DELETE A MACRO

This command deletes one or all macros from memory. (It does not affect macro files.) To delete a single macro, enter the macro name. To delete all macros, enter "*". Take care with the use of *. It will delete ALL your Macros.

16.23 DEFAULTS

Defaults at Program Startup:
Coordinate mode is Absolute
Yes/No mode is A (ask)
Macro replay count is 1.

16.24 LIMITS

Description	EMM (640K)	No EMM (below 640K)
Total number of keystrokes	4096	100

Total number of macros	56	20
Maximum nesting depth	64	5

16.25 MACRO LANGUAGE

The proceeding portion of this chapter described the creation of Macros using the cursor, the Keyboard and Menu Commands to create the Macros and record it while actually doing the intended Macro.

In some instances, it is better to use a Text Editor (PEDIT, WORD PERFECT, etc.) to create an ASCII file of the Macro. To do this requires a knowledge of the Macro language.

The following describes the elements of the macro language. Unless otherwise specified, all elements are upper/lower case insensitive.

Comment
(*XXX*) where XXX is any number of characters as long as the total including the "(*" and "*)" does not exceed 80 and everything is on one line.

F-Keys
(F1)
(F2)
(F3)
(F4)
(F5)
(F6)
(F7)
(F8)
(F9)
(F10)
F11 and F12 are not supported

Numeric Keypad Keys
(home)
(end)
(up)
(down)

(left)
(right)
(pgup)
(pgdn)
(ins)
(del)

Control Characters

If a macro must generate control characters such as Carriage Return or Tab, the following must be used.

Macro Element	Control Character	Description
(soh)	^A	
(stx)	^B	
(etx)	^C	
(eot)	^D	
(enq)	^E	
(ack))	^F	
(bel)	^G	
(bs)	^H	Backspace
(ht)	^I	Tab
(lf)	^J	Linefeed
(vt)	^K	
(ff)	^L	Formfeed
(cr)	^M	Carriage Return (Enter)
(so)	^N	
(si)	^O	
(dle)	^P	
(dc1)	^Q	
(dc2)	^R	
(dc3)	^S	
(dc4)	^T	
(nak)	^U	
(syn)	^V	
(etb)	^W	
(can)	^X	
(em)	^Y	
(sub)	^Z	
(esc)	^[	Escape

(fs)	^\	
(gs)	^]	
(rs)	^^	
(us)	^_	
(btab)	none	Back Tab

Alt, Control and Shift

Any of the above can be Alt'd, Ctrl'l or Shifted (where appropriate) with the addition of a-, c- or s- respectively. For instance Alt-F1 is (a-f1), Ctrl-F1 is (s-f1), and Shift -F1 is (s-f1).

Macro Statements

(bdef) (*desc*) (key)	begin macro definition
(edef)	end macro definition
(rstr)	return to saved menu location
(top)	return to first menu
(yn) n	set Yes/No mode (y(es), n(o), a (sk), m(acro), r(eset))
(move) x y	move cursor to (x,y)
(rel)	coordinate following next command is relative

16.26 SAMPLE MACROS

A Macro to Unconditionally Exit the System

(bdef) (*exit*) (a-x)	(*begin definition of macro alt-x*)
(yn) y	(*answer Yes to all Yes/No*)
(top)	(*go to the top*)
(f10)	(*exit*)
(edef)	(*end of definition*)

A Macro to Get to the Macro Management Menu

(bdef) (*exit*) (a-m)	(*begin definition of macro alt-m*)
(yn) a	(*ask me at yes/no*/)
(top)	(*go to the top*)
(f8)	(*Setup*)
(f3)	(*Macro*)
(edef)	(*end of definition*)

A Macro to Save a Specific Schematic File

```
(bdef) (*get mem*) (s-f1)   (*begin defn of macro shift-F1*)
(top)                       (*go to the top*)
(f1)                        (*In/Out*)
(yn) y                      (*answer yes at yes/no*)
(f2)                        (*Job Out*)
memory (cr)                 (*name of schematic file*)
(yn) r                      (*reset yes/no mode*)
(rstr)                      (*restore menu position*)
```

16.27 MODELESS COMMANDS

A somewhat different form of Macros are the Modeless Macros or Modeless Commands provided by CAD Software and contained in the software.

Although strictly speaking, these are not true Macros, since their use is to save key strokes and thus time, they are contained within this chapter on Macros.

CAD Softare has built into ***PADS-LOGIC***, a number of "time savers" that you should use. Generally, these are single key strokes programmed to perform a specific function.

The following lists these Modeless Commands and their function. They are all case insensitive. Become familiar with them.

TXX	Changes current Text Height to XX mils.
WYY	Changes current Text Width to XX mils.
GZ	Changes the current System Grid to Z mils
SHN	Changes to Sheet N
HI	Pushes DOWN through the selected Hierarchy Symbol.
HO	Pushes UP from within the Hierarchical circuit to the Hierarchical Symbol.

SU1A — Searches for Part U1A, displays the sheet where U1A is located, then centers the cursor on the White Outline of the Part.

Ctrl End — Brings back the last prompt response or command entered at the Prompt Line.

ALT0 through ALT9: Reserved for Macro Command

Zn <CR> — The Zoom will be by the factor N following <CR>
Zoom factors are:
1 = 64" complete database (64" x 64")
2 = 32"
3 = 21-23"
4 = 16"
etc.

Cn — The following macro will be replayed n times.

NOTE: Modeless Macros function at the higher levels of the Menu structure, but depending upon the Modeless Macro involved, may not function at a lower level. Example: Search (SU1A) will NOT function if in ADD PART, and a part is attached to the cursor.

APPENDIX A

PADS™-LOGIC

LIBRARY PARTS

APPENDIX A

LIBRARY PARTS

This Appendix contains an Alphanumeric listing of both the Part Types as well as the Decals shipped with ***PADS-LOGIC***. Part I contains the alphanumeric listing of the Part Types, Part II of this Appendix contains an alphanumeric listing of the Decals used in the Part Type.

CAUTION: The User is cautioned that while CAD Software, Inc. has used due diligence to insure the accuracy of the Parts Library, CAD Software, Inc. will not be responsible for any consequential damages resulting from the use of the library. We recommend that prior to using a Part for the first time that you check the Part Type information and Gate information against the Vendor's specification sheet or other data you may have.

$GND_SYMS	PCBNULL	GND: AGND: CHGND
$OSR_SYMS	PCBNULL	REFIN: REFOUT
$PWR_SYMS	PCBNULL	+5V: BUBBLE: Y
100415	DIP16	100415
100469	DIP24\400	100469
100470	DIP18	100470
100474	DIP24\400	100474
100480	DIP20	100480
100P14	DIP20	100P14
100P44	DIP20	100P44
100P88	DIP28\600	100P88
10415	DIP16	100415
10469	DIP24\400	100469
10470	DIP18	100470
10474	DIP24\400	100474
10480	DIP20	10480
10KP14	DIP24\600	100P14
10KP14J	DIP24\600	100P14
10P14	DIP20	100P14
10P44	DIP20	100P44
10P88	DIP28\600	100P88
1372	DIP14	1372
146805E2	DIP40\600	DIP38
146805E2Z	CC40\40	DIP38

146805F2	DIP28\600	DIP26
146805F2-PLCC	CC40\40	DIP26
146805F2Z	CC40\40	DIP26
146805G2	DIP40\600	DIP38
146805G2Z	CC40\40	DIP38
146805H2	DIP40\600	DIP38
146805H2Z	CC40\40	DIP38
146818	DIP24\600	DIP22
146818A	DIP24\600	DIP22
146818Z	DIP24\600	DIP20
146823	DIP40\600: CC40\40	DIP38
1468705F2	DIP28\600	DIP26
1468705G2	DIP40\600	DIP38
16164PLCC	CC68	16164
16450	DIP40\600	8250
16550	DIP40\600	16550
2130	DIP48\600	RS1024K8
2130L	CC52: CL52	RS1024K8
2147	DIP18: CLR18	2147
2148	DIP18: CLR18	2148
2149	DIP18: CLR18	2148
2150	DIP24	2150
2167	DIP20: CLR20	6167
2168	DIP20: CLR20	6168

2169	DIP20: CLR20	6168
21L4I	DIP18	21L41
21L50	DIP24	2150
24C16	DIP8	24C16
2641	DIP24\400	2641
2641A	CC28	2641
2651	DIP28\600	2651
2652	DIP40\600	2652
2652A	CC44	2652
2653	DIP16: DIP16\SO	2653
26562	DIP48\600	68562
2661	DIP28\600: CC28	2661
2670	DIP28\600	2670
2671	DIP40\600	DIP38
2672	DIP40\600	2672
2672A	CC44	2672
2673	DIP40\600	DIP38
2674	DIP40\600	2672
2674A	CC44	2672
2675	DIP40\600	2675T
2675T	DIP40\600	2675T
2677	DIP40\600	2677
2677A	CC44	2677
2681A	CC44: CC44A	2681N40

2681N24	DIP24\400	2681N24
2681N28	DIP28\600	2681N28
2681N40	DIP40\600	2681N40
2691	DIP24\300	2691
2691A	CC28	2691
2692A	CC44: CC44A	2681N40
2692N24	DIP24\400	2681N24
2692N28	DIP28\600	2681N28
2692N40	DIP40\600	2681N40
2698	DIP64\900	2698SM
2698A	CC84	2698
27128A	DIP28\600	RE16K8: RE16K8-A
27128AL	CLR32	RE16K8: RE16K8-A
2716B	DIP24\600	RE2K8
27256	DIP28\600	RE32K8: RE32K8-A
27256L	CLR32	RE32K8: RE32K8-A
2732B	DIP24\600	RE4K8
27512	DIP28\600	27512
2764A	DIP28\600	2764
2764AL	CLR32	2764
27C010	DIP32\600: CCR32	RE128K8: RE128K8-A
27C020	DIP32\600	RE256K8: RE256K8-A
27C1023	DIP32\600	RE128K8: RE128K8-A
27C1024	DIP40\600	RE64K16

27C1024L	CL44:CC44	RE64K16
27C128	DIP28\600	RE16K8:RE16K8-A
27C256	DIP28\600	RE32K8:RE32K8-A
27C256L	CLR32	RE32K8:RE32K8-A
27C256OTP	DIP28\600	RE32K8:RE32K8-A
27C512	DIP28\600	RE64K8:RE64K8-A
27C512L	CLR32	RE64K8:RE64K8-A
27C512OTP	DIP28\600	RE64K8:RE64K8-A
27C64	DIP28\600	RE8K8:RE8K8-A
27C64L	CLR32	RE8K8:RE8K8-A
27LS00	DIP16	82LS16
27LS00L	CL20	82LS16
27LS01	DIP16	82LS16
27LS01L	CL20	82LS16
27LS02	DIP16	82S25
27LS02L	CL20	82S25
27LS03	DIP16	82S25
27LS03L	CL20	82S25
27LS06	DIP16	82S25
27LS06L	CL20	82S25
27LS07	DIP16	82S25
27LS07L	CL20	82S25
27LS18	DIP16	82S123
27LS185	DIP18\300	82S185

27LS185L	CLR28: CL28	82S185
27LS18L	CL20	82S123
27LS19	DIP16	82S123
27LS191	DIP24\600	27S190
27LS191L	CLR32	27S190
27LS191S	CL28	27S190
27LS19L	CC20: CC20A: CLR20	82S123
27LS291	DIP24\600: DIP24\300	27S190
27LS291L	CLR32	27S190
27LS291S	CL28	27S190
27PS181	DIP24\600	82S181
27PS181L	CLR32	82S181
27PS181S	CL28	82S181
27PS185	DIP18	82S185
27PS185L	CLR28: CL28	RE2048K4
27PS191	DIP24\600	27S190
27PS191L	CLR32	27S190
27PS191S	CL28	27S190
27PS191U	CLR32	82S191
27PS281	DIP24\600: DIP24\300	27S180
27PS281L	CLR32	82S181
27PS281S	CL28: CC28	82S181
27PS291	DIP24\600	27S190
27PS291L	CLR32	27S190

27PS291S	CL28	27S190
27PS291U	CLR32	82S191
27PS41	DIP20	82HS195
27PS41J	CC28	82HS195
27S12	DIP16	82S130
27S12L	CL20	82S130
27S13	DIP16	82S130
27S13L	CL20: CC20: CC20A	82S130
27S15	DIP24\600	82S115
27S180	DIP24\600	27S180
27S180L	CLR32	82S181
27S180S	CL28	82S181
27S181	DIP24\600	27S180
27S181L	CLR32	82S181
27S181S	CL28: CC28	82S181
27S184	DIP18	RE2048K4
27S184L	CLR28	RE2048K4
27S184S	CL28	RE2048K4
27S185	DIP18	RE2048K4
27S185L	CLR28	RE2048K4
27S185S	CL28	RE2048K4
27S190	DIP24\600	27S190
27S190L	CLR32	27S190
27S190S	CL28	27S190

27S191	DIP24\600	27S190
27S191L	CLR32	27S190
27S191S	CL28	27S190
27S20	DIP16	82S126
27S20L	CL20	82S126
27S21	DIP16	82S126
27S21L	CL20	82S126
27S25	DIP24	27S25
27S25L	CLR32	27S25
27S25S	CL28	27S25
27S27	DIP22	27S27
27S28	DIP20; CL20	27S28
27S280	DIP24	27S180
27S280L	CLR32	82S181
27S280S	CL28; CC28	82S181
27S281	DIP24\600; DIP24	27S180
27S281L	CLR32	82S181
27S281S	CL28; CC28	82S181
27S29	DIP20; CL20	27S28
27S290	DIP24\600; DIP24	27S190
27S290L	CLR32	27S190
27S290S	CL28	27S190
27S291	DIP24\600; DIP24	27S190
27S291L	CLR32	27S190

27S291S	CL28	27S190
27S31	DIP24\600	27S31
27S31L	CLR32	27S31L
27S31S	CL28	27S31L
27S32	DIP18	27S32
27S32L	CL20	27S32
27S33	DIP18	27S32
27S33L	CL20	27S32
27S35	DIP24	27S35
27S35L	CLR32	27S35
27S35S	CL28	27S35
27S37	DIP24	27S35
27S37L	CLR32	27S35
27S37S	CL28	27S35
27S41	DIP20	27S41
27S43	DIP24\600	RE4K8
27S43L	CLR32	RE4K8
27S43S	CL28	RE4K8
27S45	DIP24\600	RE4K8
27S45L	CLR32	RE4K8
27S45S	CL28	RE4K8
27S47	DIP24\600	RE4K8
27S47L	CLR32	RE4K8
27S47S	CL28	RE4K8

27S49	DIP24\600	27S49
27S49L	CLR32	27S49
27S49S	CL28	27S49
27S51	DIP28\600	27S51
27S55	DIP24	27S55
27S65	DIP24	27S65
27S65L	CL28	27S65
27S75	DIP24	27S75
27S75L	CL28	27S75
27S85	DIP24	27S85
27S85L	CL28	27S85
27S95	DIP28\400	27S95
2817A	DIP28\600	RS2K8
2817A-LCC	CLR32	RE2K8
2817AL	CLR32	RS2K8
2864	DIP28\600	RS8K8: RS8K8-A
2864A	DIP28\600	RS8K8: RS8K8-A
2864AL	CLR32	RS8K8: RS8K8-A
2864B	DIP28\600	RS8K8: RS8K8-A
2864BL	CLR32	RS8K8: RS8K8-A
2864L	CLR32	RS8K8: RS8K8-A
28C010	DIP32\600	RS128K8: RS128K8-A
28C256	DIP28\600	RS32K8
28C256L	CLR32	RS32K8

29PL14I	DIP28\600: CL28	DIP26
2N2222	TO-18	NPN-V: NPN-VA: NPN-H: NPN-HA
2N2907	TO-18	PNP-H: PNP-V
2N2955	TO-3	PNP-V: PNP-VA: PNP-H: PNP-HA
2N3053	TO-39	NPN-V: NPN-VA: NPN-H: NPN-HA
2N3055	TO-3	NPN-V: NPN-VA: NPN-H: NPN-HA
2N4030	TO-39	NPN-V: NPN-VA: NPN-H: NPN-HA
31L01	DIP16	31L01
31L01L	CL20	31L01
32016	DIP48\600	32016
32032	CC68: PGA\68PB	32032
32081	DIP24\600	32081
32082	DIP48\600	32082
32201	DIP24\600	32201
32202	DIP40\600	32202
32440	DIP24\600	32440
32440V	CC28: CC28A	32440
32441	DIP24\600	32441
32441V	DIP24\600	32441
32442	DIP24\600	32442
32443	DIP24\600	32443
32490	DIP48\600	32490
32491	DIP24\600	32491
32492	DIP16	32492

3440A	DIP16	3440
3441A	DIP16	3440
3443A	DIP16	3440
3446A	DIP16	3446
3447	DIP24: DIP24\600	3447
3448A	DIP16	3448A
3870	DIP40\600	3870
4000B	DIP14: DIP14\SO	NOR3 INV
4000C	DIP14: DIP14\SO	NOR3 INV
4000M	DIP14: DIP14\SO	NOR3 INV
4001	DIP14	NOR2
4001B	DIP14	NOR2
4001BC	DIP14	NOR2
4001BCP	DIP14	NOR2
4001BM	DIP14	NOR2
4001C	DIP14	NOR2
4001M	DIP14	NOR2
4002	DIP14	NOR4
4002B	DIP14	NOR4
4002BC	DIP14	NOR41
4002BM	DIP14	NOR4
4002C	DIP14	NOR4
4002M	DIP14	NOR4
4006	DIP14	4006

4006B	DIP14	4006
4006BC	DIP14	4006
4006BM	DIP14	4006
4007	DIP14	4007
4007C	DIP14	4007
4007M	DIP14	4007
4007UB	DIP14	4007
4008	DIP16	4008
4008B	DIP16	4008
4008BC	DIP16	4008
4008BM	DIP16	4008
4009	DIP16	INV: INVDM
4009B	DIP16	INV: INVDM
4009C	DIP16	INV: INVDM
4009M	DIP16	INV: INVDM
4010	DIP16	INV: INVDM
40106	DIP14	INVST
40106B	DIP14: DIP14\SO	INVST
40106BC	DIP14	INVST
40106BM	DIP14	INVST
4010B	DIP16	INV: INVDM
4010C	DIP16	INV: INVDM
4010M	DIP16	INV: INVDM
4011	DIP14	NAND2

4011B	DIP14:DIP14\SO	NAND2
4011BC	DIP14	NAND2
4011BM	DIP14	NAND2
4011C	DIP14	NAND2
4011M	DIP14	NAND2
4011UB	DIP14	NAND2
4012	DIP14	NAND4
4012B	DIP14	NAND4
4012BC	DIP14	NAND4
4012BM	DIP14	NAND4
4012C	DIP14	NAND4
4012M	DIP14	NAND4
4013	DIP14	\74
4013B	DIP14	\74
4013BC	DIP14	\74
4013BM	DIP14	\74
4014	DIP16	4014
4014B	DIP16	4014
4014BC	DIP16	4014
4014BM	DIP16	4014
4015	DIP16	4015
4015B	DIP16	4015
4015BC	DIP16	4015
4015BM	DIP16	4015

4016	DIP14	4016
40160B	DIP16: DIP16\SO	\160
40160BC	DIP16: DIP16\SO	\160
40160BM	DIP16: DIP16\SO	\160
40161B	DIP16: DIP16\SO	\160
40161BC	DIP16: DIP16\SO	\160
40161BM	DIP16: DIP16\SO	\160
40162B	DIP16: DIP16\SO	\160
40162BC	DIP16: DIP16\SO	\160
40162BM	DIP16: DIP16\SO	\160
40163B	DIP16: DIP16\SO	\160
40163BC	DIP16: DIP16\SO	\160
40163BM	DIP16: DIP16\SO	\160
4016B	DIP14	4016
4016BC	DIP14	4016
4016BM	DIP14	4016
4017	DIP16	4017
40174B	DIP16: DIP16\SO	\174
40174BC	DIP16: DIP16\SO	\174
40174BM	DIP16: DIP16\SO	\174
40175B	DIP16: DIP16\SO	40175
40175BC	DIP16: DIP16\SO	40175
40175BM	DIP16: DIP16\SO	40175
4017B	DIP16	4017

4017BC	DIP16	4017
4017BM	DIP16	4017
4018	DIP16	4018
4018B	DIP16	4018
4018BC	DIP16	4018
4018BM	DIP16	4018
4019	DIP16	4019
40192B	DIP16: DIP16\SO	\192
40192BC	DIP16: DIP16\SO	\192
40192BM	DIP16: DIP16\SO	\192
40193B	DIP16: DIP16\SO	40193
40193BC	DIP16: DIP16\SO	40193
40193BM	DIP16: DIP16\SO	40193
4019B	DIP16	4019
4019BC	DIP16	4019
4019BM	DIP16	4019
4020	DIP16	4020
4020B	DIP16	4020
4020BC	DIP16	4020
4020BM	DIP16	4020
4021	DIP16	4021
4021B	DIP16	4021
4021BC	DIP16	4021
4021BM	DIP16	4021

4022	DIP16	4022
4022B	DIP16	4022
4022BC	DIP16	4022
4022BM	DIP16	4022
4023	DIP14	NAND3
4023B	DIP14	NAND3
4023BC	DIP14	NAND3
4023BCP	DIP14	NAND3
4023BM	DIP14	NAND3
4023C	DIP14	NAND3
4023M	DIP14	NAND3
4024	DIP14	4024
4024B	DIP14	4024
4024BC	DIP14	4024
4024BM	DIP14	4024
4025	DIP14	NOR3
4025B	DIP14	NOR3
4025BC	DIP14	NOR3
4025BM	DIP14	NOR3
4025C	DIP14	NOR3
4025M	DIP14	NOR3
4027	DIP16	4027
4027B	DIP16	4027
4027BC	DIP16	4027

4027BM	DIP16	4027
4028	DIP16	4028
4028B	DIP16	4028
4028BC	DIP16	4028
4028BM	DIP16	4028
4029	DIP16	4029
4029B	DIP16	4029
4029BC	DIP16	4029
4029BM	DIP16	4029
4030	DIP14	XOR
4030B	DIP14	XOR
4030C	DIP14	XOR
4030M	DIP14	XOR
4031	DIP16	4031
4031B	DIP16	4031
4031BC	DIP16	4031
4031BM	DIP16	4031
4034	DIP24\600	4034
4034B	DIP24\600	4034
4034BC	DIP24\600	4034
4034BM	DIP24\600	4034
4035	DIP16	4035
4035B	DIP16	4035
4035BC	DIP16	4035

4046BM	DIP16	4046
4047	DIP14: DIP14\SO	4047
4047B	DIP14: DIP14\SO	4047
4047BC	DIP14: DIP14\SO	4047
4047BM	DIP14: DIP14\SO	4047
4048	DIP16: DIP16\SO	4048
4048B	DIP16: DIP16\SO	4048
4048BC	DIP16: DIP16\SO	4048
4048BM	DIP16: DIP16\SO	4048
4049	DIP16	INV: INVL: INVDM: INVDML
4049B	DIP16	INV: INVL: INVDM: INVDML
4049UB	DIP16	INV: INVL: INVDM: INVDML
4049UBC	DIP16	INV: INVL: INVDM: INVDML
4049UBM	DIP16	INV: INVL: INVDM: INVDML
405	DIP48\600	405
4050	DIP16	INVNON: INVNONL
4050B	DIP16	INVNON: INVNONL
4050BC	DIP16	INVNON: INVNONL
4050BCP	DIP16	INVNON: INVNONL
4050BM	DIP16	INVNON: INVNONL
4050M	DIP16	INVNON: INVNONL
4051B	DIP16: DIP16\SO	4051
4051BC	DIP16: DIP16\SO	4051
4051BM	DIP16: DIP16\SO	4051

4052	DIP16	4052
4052B	DIP16	4052
4052BC	DIP16	4052
4052BM	DIP16	4052
4053	DIP16	4052
4053B	DIP16	4052
4053BC	DIP16	4052
4053BM	DIP16	4052
4060	DIP16	4060
4060B	DIP16	4060
4060BC	DIP16	4060
4060BM	DIP16	4060
4061B	DIP16:DIP16\SO	4060
4066	DIP14	4016
4066B	DIP14	4016
4066BC	DIP14	4016
4066BM	DIP14	4016
4068B	DIP14:DIP14\SO	NAND8
4069	DIP14	INV:INVDM
4069B	DIP14	INV:INVDM
4069UB	DIP14	INV:INVDM
4069UBC	DIP14	INV:INVDM
4069UBM	DIP14	INV:INVDM
4070	DIP14	XOR

4070B	DIP14	XOR
4070BC	DIP14	XOR
4070BM	DIP14	XOR
4071	DIP14	OR2
4071B	DIP14	OR2
4071BC	DIP14	OR2
4071BM	DIP14	OR2
4072	DIP14	OR4
4072B	DIP14	OR4
4072BC	DIP14	OR4
4072BM	DIP14	OR4
4073	DIP14	AND3
4073B	DIP14	AND3
4073BC	DIP14	AND3
4073BM	DIP14	AND3
4075	DIP14	OR3
4075B	DIP14	OR3
4075BC	DIP14	OR3
4075BM	DIP14	OR3
4076	DIP16	4076
4076B	DIP16	4076
4076BC	DIP16	4076
4076BM	DIP16	4076
4077	DIP14: DIP14\SO	XNOR

4077B	DIP14: DIP14\SO	XNOR
4078B	DIP14: DIP14\SO	NOR8
4081	DIP14	AND2
4081B	DIP14	AND2
4081BC	DIP14	AND2
4081BM	DIP14	AND2
4082	DIP14	AND4
4082B	DIP14	AND4
4082BC	DIP14	AND4
4082BM	DIP14	AND4
4089	DIP16	4089
4089B	DIP16	4089
4089BC	DIP16	4089
4089BM	DIP16	4089
4093	DIP14	NAND2: NAND2DM
4093B	DIP14	NAND2: NAND2DM
4093BC	DIP14	NAND2: NAND2DM
4093BM	DIP14	NAND2: NAND2DM
4094	DIP16	4094
4094B	DIP16	4094
4094BC	DIP16	4094
4094BM	DIP16	4094
4099	DIP16	4099
4099B	DIP16	4099

4099BC	DIP16	4099
41464	DIP18	RD64K4
4316B	DIP16:DIP16\SO	4316
4351B	DIP20:DIP20\SOL	4351
4352B	DIP20:DIP20\SOL	4352
4353B	DIP20:DIP20\SOL	4353
4511B	DIP16:DIP16\SO	4511
4512B	DIP16:DIP16\SO	4512
4518B	DIP16:DIP16\SO	\4518
4520B	DIP16:DIP16\SO	\4518
4527B	DIP16	4089
4528B	DIP16:DIP16\SO	\4538
4538B	DIP16:DIP16\SO	\4538
4539B	DIP16:DIP16\SO	\153
4555B	DIP16:DIP16\SO	4555
4556B	DIP16:DIP16\SO	4556
4724B	DIP16:DIP16\SO	4724
48416	DIP18	RD16K4
4864	DIP16	RD256K1
50256	DIP16	RD256K1
50257	DIP16	RD256K1
511000	DIP18	RD1024K1
514256	DIP20	RD256K4
6116	DIP24\600	6116

6167	DIP20	6167
6167SA	DIP20	6167
6167SL	DIP20	6167
6168	DIP20:CLR20	6168
6208	DIP24	RS64K4
62256	DIP28\600:DIP28\SOL	RS32K8
6264	DIP28\600:DIP28\SOL	RE8K8
6287	DIP22	6287
63484	DIP64\900	63484
68000	DIP64\900	68000
68000PGA	PGA\68PA	68000
68000PLCC	CC68	68000
6801	DIP40\600	DIP38
68010	DIP64\900	68000
68010A	CC68	68000
68010PGA	PGA\68PA	68000
6801U4	DIP40\600	DIP38
6802	DIP40\600	6802
6802NS	DIP40\600	6802
6803	DIP40\600	DIP38
6803E	DIP40\600	DIP38
6803U4	DIP40\600	DIP38
6804J2	DIP20	DIP18
6804P2	DIP28\600	DIP26

6805K2	DIP40\600	DIP40
6805K3	DIP40\600	DIP40
6805P2	DIP28\600	DIP26
6805P4	DIP28\600	DIP26
6805P6	DIP28\600	DIP26
6805R2	DIP40\600	DIP40
6805R3	DIP40\600	DIP38
6805S2	DIP28\600	DIP26
6805S3	DIP28\600	DIP26
6805T2	DIP28\600	6805
6805U2	DIP40\600	DIP38
6805U3	DIP40\600	DIP38
6808	DIP40\600	6802
6809	DIP40\600	DIP38
6809E	DIP40\600	DIP38
68120	DIP48\600	DIP46
68121	DIP48\600	DIP46
68154	DIP40\600	68154
68154-PLCC	CC44	68154
68154A	CC44	68154
68155	DIP40\600	68155
68155A	CC44	68155
68155PLCC	CC44	68155
68171	DIP16	68171

68172	DIP28\600	68172
68172A	CC44	68172
68172PLCC	CC44	68172
68173	DIP28\600	68173
6821	DIP40\600	DIP38
6822	DIP40\600	DIP38
6829	DIP40\600	DIP38
6835	DIP40\600	DIP38
6839	DIP24\600	6839
6840	DIP28\600	6840
68430	DIP48\600	68430
68430A	CC52	68430
68430PLCC	CC52	68430
6844	DIP40\600	DIP38
6845	DIP40\600	6845
6846	DIP40\600	6846
6847	DIP40\600	6847
6847Y	DIP40\600	6847
6850	DIP24\600	DIP22
6852	DIP24\600	6852
6854	DIP28\600	6854
68562	DIP48\600	68562
68562A	DIP48\600	68562
6859	DIP24\600	DIP22

68661	DIP28\600: CC28	2661
68681	DIP40\600	2681N40
68681-PLCC	CC44	2681N40
68681A	CC44: CC44A	2681N40
68692	DIP40\600	2681N40
68692-PLCC	CC44: CC44A	2681N40
68692A	CC44: CC44A	2681N40
68701	DIP40\600	DIP38
68701U4	DIP40\600	DIP38
68705P3	DIP28\600	DIP26
68705P5	DIP28\600	DIP26
68705R3	DIP40\600	DIP38
68705R5	DIP40\600	DIP38
68705U3	DIP40\600	DIP38
68705U5	DIP40\600	DIP38
6875	DIP16	6875
6880A	DIP16	6880A
6882A	DIP20	DIP20
6882B	DIP20	DIP20
6883	DIP40\600	6883
6885	DIP16	DIP16
6886	DIP16	DIP16
6887	DIP16	DIP16
6888	DIP16	DIP16

6889	DIP16	DIP16
6890	DIP20	6890
68HC04J2	DIP20	DIP18
68HC04J3	DIP20	DIP18
68HC04P2	DIP28\600	68HC04
68HC04P3	DIP28\600	68HC04
68HC09E	DIP40\600	68HC09
68HC34	DIP40\600	DIP38
68HC51	DIP28\600	68HC51
68HC68A1	DIP16	68HC68
68HC68R1	DIP8	68HC68R1
68HC68R2	DIP8	68HC68R1
7990	DIP48\600	7990
7992A	DIP24\300	7992
80186	PGA\68P: CC68A	DIP64
80188	PGA\68P: CC68A	DIP64
80286	PGA\68P	80286
8031	DIP40\600	8031
8031AH	DIP40\600	8031
8032AH	DIP40\600	8031
8032BH	DIP40\600	8031
8035AHL	DIP40\600	8035AHL
8035H	DIP40\600	8035AHL
8039AHL	DIP40\600	8035AHL

8039H	DIP40\600	8035AHL
8040AHL	DIP40\600	8035AHL
8041AH	DIP40\600	8041
8042AH	DIP40\600	8042
8044AH	DIP40\600	8031
8048AH	DIP40\600	8035AHL
8049AH	DIP40\600	8035AHL
8050AH	DIP40\600	8035AHL
8051	DIP40\600	8031
8051AH	DIP40\600	8031
8052	DIP40\600	8031
8052AH	DIP40\600	8031
8052BH	DIP40\600	8031
8080A	DIP40\600	8088
8085AH	DIP40\600	8085AH
8086	DIP40\600	8086
8088	DIP40\600	8088
8094	DIP48\600	8094
8095	DIP48\600	8094
8095BH	DIP48\600	8094
8095CH	DIP48\600	8094
8096	PGA\68P	8096
8096BH	PGA\68P	8096
8097	PGA\68P	8096

8097BH	PGA\68P	8096
80C252	DIP40\600	80C252
80C31BH	DIP40\600	8031
80C32AH	DIP40\600	8031
80C39	DIP40\600	8035AHL
80C51BH	DIP40\600	80C252
80C52AH	DIP40\600	8031
8152A	DIP48\600	8152
8152AL	CL52	8152
8153A	DIP48\600	8152
8153AL	CL52	8152
8155H	DIP40\600	8155H
8156H	DIP40\600	8155H
8185	DIP18	8185
8203	DIP40\600	8203
8206	PGA\68P	8206
82064	DIP40\600	82064
8207	PGA\68P	8207
82072	DIP40\600	82072
82188	DIP28\600	82188
8224	DIP16	8224
8228	DIP28\600	8228
82284	DIP18	DIP16
82288	DIP20	DIP18

8231A	DIP24\600	8231
8237A	DIP40\600	8237A
8238	DIP28\600	8228
8243	DIP24\600	8243
8250A	DIP40\600	8250
8251A	DIP28\600	8251A
8253	DIP24\600	8253
8254	DIP24\600	8253
8255A	DIP40\600	8255A
8256AH	DIP40\600	8256AH
8257	DIP40\600	8257
8259A	DIP28\600	8259A
82716	PGA\68P: CC68A	82716
8272A	DIP40\600	8272A
82730	PGA\68P	82730
8275H	DIP40\600	8275
8276H	DIP40\600	8276
82786	PGA\88P	82786
8279	DIP40\600	8279
8284A	DIP18	8284
8288	DIP20	8288
8289	DIP20	8289
82C08	DIP48\600	82C08
82C55A	DIP40\600	8255A

82C59A	DIP28\600	DIP26
82HS195	DIP20	82HS195
82HS321	DIP24\600	82HS321
82HS641	DIP24\600	82HS641
82LS16	DIP16	82LS16
82S115	DIP24\600	82S115
82S123	DIP16	82S123
82S126	DIP16	82S126
82S129	DIP16	82S129
82S130	DIP16	82S130
82S131	DIP16	82S130
82S181	DIP24\600	82S181
82S185	DIP18	82S185
82S191	DIP24\600	82S191
82S191CN3	DIP24	82S191
82S23	DIP16	82S123
82S25	DIP16	82S25
8340	DIP24\600	8340
8341	DIP24\600	8341
8342	DIP24\600	8342
8343	DIP24\600	8343
8344AH	DIP40\600	8031
8394	DIP48\600	8094
8395	DIP48\600	8094

8395BH	DIP48\600	8094
8396	PGA\68P	8096
8396BH	PGA\68P	8096
8397	PGA\68P	8096
8397BH	PGA\68P	8096
83C154	DIP40\600	DIP38
83C452P	PGA\68P	DIP64
8741AH	DIP40\600	8041
8742AH	DIP40\600	DIP38
8744H	DIP40\600	8031
8748H	DIP40\600	8035AHL
8749H	DIP40\600	8035AHL
8751BH	DIP40\600	8031
8751H	DIP40\600	8031
8752BH	DIP40\600	8031
8755A	DIP40\600	DIP38
8795BH	DIP48\600	DIP44
87C256	DIP28\600	RE32K8
87C452P	PGA\68P	DIP64
87C51	DIP40\600	8031
87C51-PLCC	CC44: CC44A	8031
87C51FA	DIP40\600	DIP38
87C64	DIP28\600	RE8K8
87C652	DIP40\600	DIP38

8X305	DIP50\900	8X305
8X310	DIP40\600	8X310
8X320	DIP40\600	8X320
8X372	DIP24\400	8X372
8X376	DIP24\400	8X372
9044	DIP18	9044
90C644	DIP24\400	90C644
9114	DIP18	9114
9124	DIP18	9114
9128	DIP24\600	9128
9151	DIP24	9151
93L412	DIP22\400	93L412
93L412L	CLR28	93L412
93L415	DIP16	93L415
93L415L	CLR20	93L415
93L422	DIP22\400	93L412
93L422L	CLR28	93L412
93L425	DIP16	93L415
93L425L	CLR20	93L415
93L469	DIP24	93L469
9864	DIP28\600	9864
9864L	CLR32	9864
99C10	DIP24\400	99C10
99C164	DIP22	99C164

99C19	DIP28\600	99C19
99C328	DIP28\600	99C328
99C416	DIP40\600	99C416
99C58	DIP24	99C58
99C58L	CLR28	99C58
99C59	DIP24	99C58
99C59L	CLR28	99C58
99C60	DIP24	99C60
99C60L	CLR28	99C60
99C641	DIP22: CLR22	99C641
99C89	DIP28\600	99C89
99CL68	DIP20	99CL68
99CL88	DIP28\600	99C88
99CL88L	CLR32	99C88
A8096BH	PGA\68P	DIP64
A8097BH	PGA\68P	DIP64
A8796BH	PGA\68P	DIP64
A8797BH	PGA\68P	DIP64
ACOMPD	DIP8: DIP8\SO	AOPAMP: AOPAMPM
ACOMPQ	DIP14: DIP14\SO	AOPAMP: AOPAMPM
ADARL-8	DIP18	ADARL-8
AM29818	DIP24	\29818
AM29818-CC	CC28	\29818
AM29821	DIP24	\29821

AM29821-CC	CC28	\29821
AM29822	DIP24	\29822
AM29822-CC	CC28	\29822
AM29823	DIP24	\29823
AM29823-CC	CC28	\29823
AM29824	DIP24	\29824
AM29824-CC	CC28	\29824
AM29825	DIP24	\29825
AM29825-CC	CC28	\29825
AM29826	DIP24	\29826
AM29826-CC	CC28	\29826
AM29827	DIP24	\29827
AM29828	DIP24	\29828
AM29861	DIP24	\29861
AM29862	DIP24	\29862
AM29863	DIP24	\29863
AM29864	DIP24	\29864
AOPAMPD	DIP8:DIP8\SO	AOPAMP:AOPAMPM
AOPAMPQ	DIP14:DIP14\SO	AOPAMP:AOPAMPM
BAW62	DO7:DO35:DO41	DIO-H:DIO-V
BC548	TO-92	NPN-V:NPN-VA:NPN-H:NPN-HA
BC558	TO-92	PNP-V:PNP-VA:PNP-H:PNP-HA
BC639	TO-92	NPN-V:NPN-VA:NPN-H:NPN-HA
BC640	TO-92	PNP-V:PNP-VA:PNP-H:PNP-HA

BF199	TO-92	NPN-V; NPN-VA; NPN-H; NPN-HA
BF245	TO-92	JFETN-V
BF926	TO-92	PNP-V; PNP-VA; PNP-H; PNP-HA
BSJ174	TO-92	JFETP-V
BST100	TO-92	MFETP-V
BST72	TO-92	MFETN-V
BT139	TO-220	TRIAC-V
BT149-E	TO-92	SCR-V
BT149-M	TO-92	SCR-V
BT152	TO-220	SCR-V
BUZ71	TO-220	MFETN-V
BZT03	DO7; DO35; DO41	DIOZ-H; DIOZ-V
BZV85	DO7; DO35; DO41	DIOZ-H; DIOZ-V
BZX79	DO7; DO35; DO41	DIOZ-H; DIOZ-V
CAP	DCAP\SR21	CAP; CAPA; CAPH; CAPAH
CAP1	CAP1	CAP; CAPA; CAPH; CAPAH
CAP1\5G	CAP1	CAP\5G; CAPA\5G; CAPH\5G; CAPAH\5G
CAPSMD	CC1206; CC1210; CC1812	CAP; CAPA; CAPH; CAPAH
CAPSMD	CC1206; CC1210; CC1812	CAP\5G; CAPA\5G; CAPH\5G; CAPAH\5G
CAP\5G	DCAP\SR21	CAP\5G; CAPA\5G; CAPH\5G; CAPAH\5G
CAP\CR08	CAP\CR08	CAP; CAPA; CAPH; CAPAH
CAP\CR08\5G	CAP\CR08	CAP\5G; CAPA\5G; CAPH\5G; CAPAH\5G
CAP\MA10	CAP\MA10	CAP; CAPA; CAPH; CAPAH
CAP\MA10\5G	CAP\MA10	CAP\5G; CAPA\5G; CAPH\5G; CAPAH\5G

CAP\MA20	CAP\MA20	CAP: CAPA: CAPH: CAPAH
CAP\MA20\5G	CAP\MA20	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MA30	CAP\MA30	CAP: CAPA: CAPH: CAPAH
CAP\MA30\5G	CAP\MA30	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MA40	CAP\MA40	CAP: CAPA: CAPH: CAPAH
CAP\MA40\5G	CAP\MA40	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MA50	CAP\MA50	CAP: CAPA: CAPH: CAPAH
CAP\MA50\5G	CAP\MA50	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MA60	CAP\MA60	CAP: CAPA: CAPH: CAPAH
CAP\MA60\5G	CAP\MA60	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MR04	CAP\MR04	CAP: CAPA: CAPH: CAPAH
CAP\MR04\5G	CAP\MR04	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MR05	CAP\MR05	CAP: CAPA: CAPH: CAPAH
CAP\MR05\5G	CAP\MR05	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MR06	CAP\MR06	CAP: CAPA: CAPH: CAPAH
CAP\MR06\5G	CAP\MR06	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MR07	CAP\MR07	CAP: CAPA: CAPH: CAPAH
CAP\MR07\5G	CAP\MR07	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CAP\MR08	CAP\MR08	CAP: CAPA: CAPH: CAPAH
CAP\MR08\5G	CAP\MR08	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CC05	CAP\MR05	CAP+: CAPA+: CAPH+: CAPAH+
CC05\5G	CAP\MR05	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CC06	CAP\MR06	CAP+: CAPA+: CAPH+: CAPAH+
CC06\5G	CAP\MR06	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+

CC07	CAP\MR07	CAP+: CAPA+: CAPH+: CAPAH+
CC07\5G	CAP\MR07	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CC08	CAP\MR08	CAP+: CAPA+: CAPH+: CAPAH+
CC08\5G	CAP\MR08	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CC09	CAP\MR04	CAP+: CAPA+: CAPH+: CAPAH+
CC09\5G	CAP\MR04	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CC20	CC20	GEN\DIP20
CC20A	CC20A	GEN\DIP20
CC28	CC28	GEN\DIP28
CC28A	CC28A	GEN\DIP28
CC40\40	CC40\40	GEN\DIP40
CC44	CC44	GEN\DIP44
CC44A	CC44A	GEN\DIP44
CC52	CC52	GEN\DIP52
CC68	CC68	GEN\DIP68
CC68A	CC68A	GEN\DIP68
CC84	CC84	GEN\DIP84
CC84A	CC84A	GEN\DIP84
CCR05	CAP\MR05	CAP+: CAPA+: CAPH+: CAPAH+
CCR05\5G	CAP\MR05	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR06	CAP\MR06	CAP+: CAPA+: CAPH+: CAPAH+
CCR06\5G	CAP\MR06	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR07	CAP\MR07	CAP+: CAPA+: CAPH+: CAPAH+
CCR07\5G	CAP\MR07	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+

CCR08	CAP\MR08	CAP+: CAPA+: CAPH+: CAPAH+
CCR08\5G	CAP\MR08	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR09	CAP\MR04	CAP+: CAPA+: CAPH+: CAPAH+
CCR09\5G	CAP\MR04	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR18	CCR18	GEN\DIP18
CCR32	CCR32	GEN\DIP32
CCR75	CAP\MA10	CAP+: CAPA+: CAPH+: CAPAH+
CCR75\5G	CAP\MA10	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR76	CAP\MA20	CAP+: CAPA+: CAPH+: CAPAH+
CCR76\5G	CAP\MA20	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR77	CAP\MA40	CAP+: CAPA+: CAPH+: CAPAH+
CCR77\5G	CAP\MA40	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR78	CAP\MA50	CAP+: CAPA+: CAPH+: CAPAH+
CCR78\5G	CAP\MA50	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CCR79	CAP\MA60	CAP+: CAPA+: CAPH+: CAPAH+
CCR79\5G	CAP\MA60	CAP\5G+: CAPA\5G+: CAPH\5G+: CAPAH\5G+
CD4015	DIP16: DIP16\SO	4015
CD4017	DIP16: DIP16\SO	4017
CD4020	DIP16: DIP16\SO	4040
CD4022	DIP16: DIP16\SO	\4022
CD4024	DIP14: DIP14\SO	\4024
CD4046	DIP16: DIP16\SO	\4046
CD4049	DIP16: DIP16\SO	INV: INVL: INVDM: INVDML
CD4051	DIP16: DIP16\SO	4051

CD4052	DIP16: DIP16\SO	4052
CD4053	DIP16: DIP16\SO	\4053
CD4060	DIP16: DIP16\SO	\4060
CD4066	DIP14: DIP14\SO	4066: 4066V
CD4075	DIP14: DIP14\SO	OR3
CD4094	DIP16: DIP16\SO	4094
CK05	CAP\MR05	CAP: CAPA: CAPH: CAPAH
CK05\5G	CAP\MR05	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CK06	CAP\MR06	CAP: CAPA: CAPH: CAPAH
CK06\5G	CAP\MR06	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CK12	CAP\MA10	CAP: CAPA: CAPH: CAPAH
CK12\5G	CAP\MA10	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CK13	CAP\MA20	CAP: CAPA: CAPH: CAPAH
CK13\5G	CAP\MA20	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CK14	CAP\MA40	CAP: CAPA: CAPH: CAPAH
CK14\5G	CAP\MA40	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CK15	CAP\MA50	CAP: CAPA: CAPH: CAPAH
CK15\5G	CAP\MA50	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CK16	CAP\MA60	CAP: CAPA: CAPH: CAPAH
CK16\5G	CAP\MA60	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CKR04	CAP\MR04	CAP: CAPA: CAPH: CAPAH
CKR04\5G	CAP\MR04	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CKR05	CAP\MR05	CAP: CAPA: CAPH: CAPAH
CKR05\5G	CAP\MR05	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G

CKR06	CAP\MR06	CAP: CAPA: CAPH: CAPAH
CKR06\5G	CAP\MR06	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CKR08	CAP\CR08	CAP: CAPA: CAPH: CAPAH
CKR08\5G	CAP\CR08	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CKR11	CAP\MA10	CAP: CAPA: CAPH: CAPAH
CKR11\5G	CAP\MA10	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CKR12	CAP\MA20	CAP: CAPA: CAPH: CAPAH
CKR12\5G	CAP\MA20	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CKR14	CAP\MA40	CAP: CAPA: CAPH: CAPAH
CKR14\5G	CAP\MA40	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CKR15	CAP\MA50	CAP: CAPA: CAPH: CAPAH
CKR15\5G	CAP\MA50	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CKR16	CAP\MA60	CAP: CAPA: CAPH: CAPAH
CKR16\5G	CAP\MA60	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
CL20	CL20	GEN\DIP20
CL25	CL25	CAP: CAPA: CAPH: CAPAH
CL28	CL28	GEN\DIP28
CL44	CL44	GEN\DIP44
CL52	CL52	GEN\DIP52
CLR18	CLR18	GEN\DIP18
CLR20	CLR20	GEN\DIP20
CLR28	CLR28	GEN\DIP28
CLR32	CLR32	GEN\DIP32
CON\SIP\1P	SIP\1P	EXTIN: EXTOUT

CON\SIP\10P	SIP\10P	EXTIN: EXTOUT
CON\SIP\11P	SIP\11P	EXTIN: EXTOUT
CON\SIP\12P	SIP\12P	EXTIN: EXTOUT
CON\SIP\13P	SIP\13P	EXTIN: EXTOUT
CON\SIP\14P	SIP\14P	EXTIN: EXTOUT
CON\SIP\15P	SIP\15P	EXTIN: EXTOUT
CON\SIP\16P	SIP\16P	EXTIN: EXTOUT
CON\SIP\17P	SIP\17P	EXTIN: EXTOUT
CON\SIP\18P	SIP\18P	EXTIN: EXTOUT
CON\SIP\19P	SIP\19P	EXTIN: EXTOUT
CON\SIP\2P	SIP\2P	EXTIN: EXTOUT
CON\SIP\20P	SIP\20P	EXTIN: EXTOUT
CON\SIP\3P	SIP\3P	EXTIN: EXTOUT
CON\SIP\4P	SIP\4P	EXTIN: EXTOUT
CON\SIP\5P	SIP\5P	EXTIN: EXTOUT
CON\SIP\6P	SIP\6P	EXTIN: EXTOUT
CON\SIP\7P	SIP\7P	EXTIN: EXTOUT
CON\SIP\8P	SIP\8P	EXTIN: EXTOUT
CON\SIP\9P	SIP\9P	EXTIN: EXTOUT
C\0805	CC0805	CAP: CAPA: CAPH: CAPAH
C\0805\5G	CC0805	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
C\1206	CC1206	CAP: CAPA: CAPH: CAPAH
C\1206\5G	CC1206	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
C\1210	CC1210	CAP: CAPA: CAPH: CAPAH

C\1210\5G	CC1210	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
C\1812	CC1812	CAP: CAPA: CAPH: CAPAH
C\1812\5G	CC1812	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
DCAP1	DCAP1	CAP: CAPA: CAPH: CAPAH
DCAP1\5G	DCAP1	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
DCAP\SR15	DCAP\SR15	CAP: CAPA: CAPH: CAPAH
DCAP\SR15\5G	DCAP\SR15	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
DCAP\SR20	DCAP\SR20	CAP: CAPA: CAPH: CAPAH
DCAP\SR20\5G	DCAP\SR20	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
DCAP\SR21	DCAP\SR21	CAP: CAPA: CAPH: CAPAH
DCAP\SR21\5G	DCAP\SR21	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
DCAP\SR26	DCAP\SR26	CAP: CAPA: CAPH: CAPAH
DCAP\SR26\5G	DCAP\SR26	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
DCAP\SR27	DCAP\SR27	CAP: CAPA: CAPH: CAPAH
DCAP\SR27\5G	DCAP\SR27	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
DCON25F	DCON25F	EXTIN: EXTOUT
DCON25M	DCON25M	EXTIN: EXTOUT
DCON9F	DCON9F	EXTIN: EXTOUT
DCON9M	DCON9M	EXTIN: EXTOUT
DIODE	DO7: DO35: DO41	DIO-H: DIO-V
DIODEZ	DO7: DO35: DO41	DIOZ-H: DIOZ-V
DIP10	DIP10	DIP10
DIP14	DIP14	GEN\DIP14
DIP14\SO	DIP14\SO	GEN\DIP14

DIP16	DIP16	GEN\DIP16
DIP16\SO	DIP16\SO	GEN\DIP16
DIP16\SOL	DIP16\SOL	GEN\DIP16
DIP18	DIP18	GEN\DIP18
DIP20	DIP20	GEN\DIP20
DIP20\SOL	DIP20\SOL	GEN\DIP20
DIP22	DIP22	GEN\DIP22
DIP22\300	DIP22\300	GEN\DIP22
DIP22\400	DIP22\400	GEN\DIP22
DIP24	DIP24	GEN\DIP24
DIP24\300	DIP24\300	GEN\DIP24
DIP24\400	DIP24\400	GEN\DIP24
DIP24\600	DIP24\600	GEN\DIP24
DIP24\SOL	DIP24\SOL	GEN\DIP24
DIP28\600	DIP28\600	GEN\DIP28
DIP28\SOL	DIP28\SOL	GEN\DIP28
DIP32\600	DIP32\600	GEN\DIP32
DIP4(14)	DIP4(14)	GEN\DIP4
DIP40\600	DIP40\600	GEN\DIP40
DIP48\600	DIP48\600	GEN\DIP48
DIP50\900	DIP50\900	GEN\DIP50
DIP52\600	DIP52\600	GEN\DIP52
DIP52\900	DIP52\900	GEN\DIP52
DIP6	DIP6	GEN\DIP6

DIP64\900	DIP64\900	GEN\DIP64	
DIP8	DIP8	GEN\DIP8	
DIP8\300	DIP8\300	GEN\DIP8	
DIP8\SO	DIP8\SO	GEN\DIP8	
DO35	DO35	DIO-H: DIO-V	
DO41	DO41	DIO-H: DIO-V	
DO7	DO7	DIO-H: DIO-V	
GAL20V8	DIP24	PAL20V8	
HEADER10	HEADER10	EXTIN: EXTOUT	
HEADER20	HEADER20	EXTIN: EXTOUT	
HEADER34	HEADER34	EXTIN: EXTOUT	
ICM7555	DIP8: DIP8\SO	NE555	
ICM7555-SO	DIP8\SO	NE555	
LM324	DIP14: DIP14\SO	AOPAMP: AOPAMPM	AOPAMPPN
LM339	DIP14: DIP14\SO	AOPAMP: AOPAMPM	AOPAMPPN
LM358	DIP8: DIP8\SO	AOPAMP: AOPAMPM	AOPAMPPN
LM3900	DIP14: DIP14\SO	AOPAMP: AOPAMPM	AOPAMPPN
LM741	DIP8: DIP8\SO	741	
MA10	CAP\MA10	CAP: CAPA: CAPH: CAPAH	
MA10\5G	CAP\MA10	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G	
MA20	CAP\MA20	CAP: CAPA: CAPH: CAPAH	
MA20\5G	CAP\MA20	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G	
MA30	CAP\MA30	CAP: CAPA: CAPH: CAPAH	
MA30\5G	CAP\MA30	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G	

MA40	CAP\MA40	CAP: CAPA: CAPH: CAPAH
MA40\5G	CAP\MA40	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MA50	CAP\MA50	CAP: CAPA: CAPH: CAPAH
MA50\5G	CAP\MA50	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MA60	CAP\MA60	CAP: CAPA: CAPH: CAPAH
MA60\5G	CAP\MA60	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC10	CAP\MA10	CAP: CAPA: CAPH: CAPAH
MC10\5G	CAP\MA10	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC20	CAP\MA40	CAP: CAPA: CAPH: CAPAH
MC20\5G	CAP\MA40	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC50	CAP\MR05	CAP: CAPA: CAPH: CAPAH
MC50\5G	CAP\MR05	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC54	CAP\MR04	CAP: CAPA: CAPH: CAPAH
MC54\5G	CAP\MR04	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC60	CAP\MR06	CAP: CAPA: CAPH: CAPAH
MC60\5G	CAP\MR06	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC70	CAP\MA20	CAP: CAPA: CAPH: CAPAH
MC70\5G	CAP\MA20	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC78	CAP\MA30	CAP: CAPA: CAPH: CAPAH
MC78\5G	CAP\MA30	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC85	CAP\MA50	CAP: CAPA: CAPH: CAPAH
MC85\5G	CAP\MA50	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC87	CAP\MA60	CAP: CAPA: CAPH: CAPAH
MC87\5G	CAP\MA60	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G

MC95	CAP\MR07	CAP: CAPA: CAPH: CAPAH
MC95\5G	CAP\MR07	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MC96	CAP\MR08	CAP: CAPA: CAPH: CAPAH
MC96\5G	CAP\MR08	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MOIRE	MOIRE	
MR04	CAP\MR04	CAP: CAPA: CAPH: CAPAH
MR04\5G	CAP\MR04	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MR05	CAP\MR05	CAP: CAPA: CAPH: CAPAH
MR05\5G	CAP\MR05	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MR06	CAP\MR06	CAP: CAPA: CAPH: CAPAH
MR06\5G	CAP\MR06	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MR07	CAP\MR07	CAP: CAPA: CAPH: CAPAH
MR07\5G	CAP\MR07	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MR08	CAP\MR08	CAP: CAPA: CAPH: CAPAH
MR08\5G	CAP\MR08	CAP\5G: CAPA\5G: CAPH\5G: CAPAH\5G
MTHOLE1	MTHOLE1	
MTHOLE2	MTHOLE2	
N8031	CC44: CC44A	8031
N8031AH	CC44: CC44A	8031
N8035AHL	CC44: CC44A	8035AHL
N8035H	CC44: CC44A	8035AHL
N8039AHL	CC44: CC44A	8035AHL
N8039H	CC44: CC44A	8035AHL
N8040AHL	CC44: CC44A	8035AHL

N8044AH	CC44: CC44A	8031
N8048AH	CC44: CC44A	8035AHL
N8048AHL	CC44: CC44A	8035AHL
N8049AH	CC44: CC44A	8035AHL
N8050AH	CC44: CC44A	8035AHL
N8051	CC44: CC44A	8031
N8051AH	CC44: CC44A	8031
N8052AH	CC44: CC44A	8031
N8096	CC68	8096
N8096BH	CC68	8096
N8097BH	CC68: PGA\68PB	DIP64
N80C196	CC68: PGA\68PB	DIP64
N80C31BH	CC44: CC44A	8031
N80C51BH	CC44: CC44A	8031
N8344AH	CC44: CC44A	8031
N8744H	CC44: CC44A	8031
N8748H	CC44: CC44A	8035AHL
N8749H	CC44: CC44A	8035AHL
N8751H	CC44: CC44A	8031
N87C51	CC44: CC44A	8031
NE5230	DIP8	NE5230
NE5512	DIP8	AOPAMP: AOPAMPM AOPAMPPN
NE5512-SO	DIP8\SO	AOPAMP: AOPAMPM AOPAMPPN
NE5514	DIP14	AOPAMP AOPAMPPN

NE5514-SO	DIP16\SOL	AOPAMP: AOPAMPM AOPAMPPN
NE5532	DIP8	AOPAMP: AOPAMPM AOPAMPPN
NE5532-SO	DIP16\SOL	AOPAMP: AOPAMPM AOPAMPPN
NE555	DIP8: DIP8\SO	NE555
NE555-SO	DIP8\SO	NE555
NE567	DIP8	NE567
P8031AH	DIP40\600	8031
P8041AH	DIP40\600	8041
P8042AH	DIP40\600	8042
P8048	DIP40\600	8035AHL
P8050	DIP40\600	8035AHL
P8050AH	DIP40\600	8035AHL
P8051AH	DIP40\600	8031
P80C31AH	DIP40\600	8031
P80C48	DIP40\600	8035AHL
P80C49	DIP40\600	8035AHL
P80C51AH	DIP40\600	8031
P80C652	DIP40\600	DIP38
P83C652	DIP40\600	DIP38
P8748	DIP40\600	8035AHL
P8749	DIP40\600	8035AHL
P87C51AH	DIP40\600	8031
P87C51FA	DIP40\600	8031
P87C652	DIP40\600	DIP38

PAL10020EG8	DIP24	10H20
PAL10020EG8L	CC28: CL28: CC28A	10H20
PAL10020EV8	DIP24	10H20
PAL10020EV8L	CC28: CL28: CC28A	10H20
PAL10H20EG8	DIP24	10H20
PAL10H20EG8L	CC28: CL28: CC28A	10H20
PAL10H20EV8	DIP24	10H20
PAL10H20EV8L	CC28: CL28: CC28A	10H20
PAL16H2	DIP20: CC20: CC20A: CL20	16H2
PAL16H8	DIP20: CC20: CC20A: CL20	16H8
PAL16HD8	DIP20: CC20: CC20A: CL20	16HD8
PAL16L2	DIP20: CC20: CC20A: CL20	16L2
PAL16L8	DIP20: CC20: CC20A: CL20	16L8
PAL16LD8	DIP20: CC20: CC20A: CL20	16LD8
PAL16P2	DIP20: CC20: CC20A: CL20	16P2
PAL16P8	DIP20: CC20: CC20A: CL20	16L8
PAL16R4	DIP20: CC20: CC20A: CL20	16R4
PAL16R6	DIP20: CC20: CC20A: CL20	16R6
PAL16R8	DIP20: CC20: CC20A: CL20	16R8
PAL16RP4	DIP20: CC20: CC20A: CL20	16R4
PAL16RP6	DIP20: CC20: CC20A: CL20	16R6
PAL16RP8	DIP20: CC20: CC20A: CL20	16R8
PAL18P8	DIP20: CC20: CC20A: CL20	18P8
PAL20H8	DIP24\600	20H8

PAL20L10	DIP24: DIP24\SOL	20L10
PAL20L10L	CC28: CC28A: CL28	20L10
PAL20L8	DIP24: DIP24\SOL	20L8
PAL20L8L	CC28: CC28A: CL28	20L8
PAL20R4	DIP24: DIP24\SOL	20R4
PAL20R4L	CC28: CC28A: CL28	20R4
PAL20R6	DIP24: DIP24\SOL	20R6
PAL20R6L	CC28: CC28A: CL28	20R6
PAL20R8	DIP24: DIP24\SOL	20R8
PAL20R8L	CC28: CC28A: CL28	20R8
PAL20RP10	DIP24: DIP24\SOL	20RP10
PAL20RP10L	CC28: CC28A: CL28	20RP10
PAL20RP4	DIP24: DIP24\SOL	20RP4
PAL20RP4L	CC28: CC28A: CL28	20RP4
PAL20RP6	DIP24: DIP24\SOL	20RP6
PAL20RP6L	CC28: CC28A: CL28	20RP6
PAL20RP8	DIP24: DIP24\SOL	20RP8
PAL20RP8L	CC28: CC28A: CL28	20RP8
PAL20XRP10	DIP24: DIP24\SOL	20RP10
PAL20XRP10L	CC28: CC28A: CL28	20RP10
PAL20XRP4	DIP24: DIP24\SOL	20RP4
PAL20XRP4L	CC28: CC28A: CL28	20RP4
PAL20XRP6	DIP24: DIP24\SOL	20RP6
PAL20XRP6L	CC28: CC28A: CL28	20RP6

PAL20XRP8	DIP24: DIP24\SOL	20RP8
PAL20XRP8L	CC28: CC28A: CL28	20RP8
PAL22P10	DIP24: DIP24\SOL	22P10
PAL22P10L	CC28: CC28A: CL28	22P10
PAL22V10	DIP24: DIP24\SOL	22V10
PAL22V10L	CC28: CC28A: CL28	22V10
PAL22XP10	DIP24: DIP24\SOL	22XP10
PAL22XP10L	CC28: CC28A: CL28	22XP10
PAL23S8	DIP20	23S8
PALHCT29M16	DIP24	29M16
PALHCT29M16L	CC28: CC28A: CL28	29M16
PGA\132P	PGA\132P	GEN\DIP132
PGA\68P	PGA\68P: CC68A: PGA\68PC	GEN\DIP68
PGA\68PA	PGA\68PA	GEN\DIP68
PGA\68PB	PGA\68PB: CC68	GEN\DIP68
PGA\68PC	PGA\68PC: CC68	GEN\DIP68
PGA\84P	PGA\84P	GEN\DIP84
PGA\88P	PGA\88P	GEN\DIP88
PH2955T	TO-220	NPN-V: NPN-VA: NPN-H: NPN-HA
PH3055T	TO-220	NPN-V: NPN-VA: NPN-H: NPN-HA
PLC153	DIP20: CC20: CC20A	PLC153
PLC20V8	DIP24	20V8
PLC473	DIP24	PLC473
PLHS153	DIP20: CC20: CC20A	PLC153

PLHS18P8A	DIP20: CC20: CC20A	18P8
PLHS473	DIP24	PLC473
PLHS473A	DIP24	PLC473
PLHS501A	CC52	PLHS501
PLHS502A	CC52	PLHS502
PLMC153	DIP20: CC20: CC20A	PLC153
PLS100	DIP28\600: CC28: CC28A	PLS100
PLS101	DIP28\600: CC28: CC28A	PLS100
PLS103	DIP28\600: CC28: CC28A	PLS103
PLS105	DIP28\600: CC28: CC28A	PLS105
PLS105A	DIP28\600: CC28: CC28A	PLS105
PLS151	DIP20: CC20: CC20A	PLS151
PLS153	DIP20: CC20: CC20A	PLC153
PLS153A	DIP20: CC20: CC20A	PLS153
PLS155	DIP20: CC20: CC20A	PLS155
PLS157	DIP20: CC20: CC20A	PLS157
PLS159	DIP20: CC20: CC20A	PLS159
PLS159A	DIP20: CC20: CC20A	PLS159
PLS161	DIP24	PLS161
PLS162	DIP24	PLS162
PLS163	DIP24	PLS163
PLS167	DIP24	PLS167
PLS167A	DIP24	PLS167
PLS168	DIP24	PLS168

PLS168A	DIP24	PLS168
PLS173	DIP24	PLS173
PLS173-CC	CC28: CC28A	PLS173
PLS173A	CC28: CC28A: CL20	PLS173
PLS179	DIP24	PLS179
PLS179A	DIP24	PLS179
PLUS153	DIP20: CC20: CC20A	PLC153
PLUS173	DIP24	PLS173
PLUS173A	DIP24	PLS173
PLUS405A	DIP28\600	PLUS405
PLUS405A-CC	CC28: CC28A	PLUS405
PN2222	TO-92	NPN-V: NPN-VA: NPN-H: NPN-HA
PN2907	TO-92	PNP-V: PNP-VA: PNP-H: PNP-HA
R1/2W	R1/2W	RESZ-H: RESZ-V: RESB-H: RESB-V
R1/4W	R1/4W: R1/4WA: R1/4WB	RESZ-H: RESZ-V: RESB-H: RESB-V
R1/8W	R1/8W: R1/8WA: R1/8WB	RESZ-H: RESZ-V: RESB-H: RESB-V
R1W	R1W	RESZ-H: RESZ-V: RESB-H: RESB-V
R2W	R2W	RESZ-H: RESZ-V: RESB-H: RESB-V
R3W	R3W	RESZ-H: RESZ-V: RESB-H: RESB-V
RDIP147RES	DIP14	RESZ-H: RESZ-V: RESB-H: RESB-V
RDIP168RES	DIP16	RESZ-H: RESZ-V: RESB-H: RESB-V
RES	R1/8W: R1/8WA	RESZ-H: RESZ-V: RESB-H: RESB-V
RESSMD-1206	CC1206	RESZ-H: RESZ-V: RESB-H: RESB-V
RLY-DPDT	PCB-NULL	RLY-DPDT

RLY-MZ-SPDT	RLY-MZ	RLY-SPDT
RLY-RLY11-DPDT	PCB-NULL	RLY-DPDT
RLY-RLY11-SPDT	PCB-NULL	RLY-SPDT
RLY-RY-DPDT	RLY-RY	RLY-SW-SPDT: RLY-COIL
RLY-SPDT	RLY-MZ	RLY-SPDT
RSIP10P5R	SIP\10P	RESZ-H: RESZ-V: RESB-H: RESB-V
RSIP10P9R	SIP\10P	RESZ-H: RESZ-V: RESB-H: RESB-V: RESZ-H1P: RESZ-V1P: RESB-H1P: RESB-V1P
RSIP10P9R+5	SIP\10P	RESZ-H5: RESZ-V5: RESB-H5: RESB-V5
RSIP10P9RG	SIP\10P	RESZ-HG: RESZ-VG: RESB-HG: RESB-VG
RSIP8P4R	SIP\8P	RESZ-H: RESZ-V: RESB-H: RESB-V
RSIP8P7R	SIP\8P	RESZ-H: RESZ-V: RESB-H: RESB-V: RESZ-H1P: RESZ-V1P: RESB-H1P: RESB-V1P
RSIP8P7R+5	SIP\8P	RESZ-H5: RESZ-V5: RESB-H5: RESB-V5
RSIP8P7RG	SIP\8P	RESZ-HG: RESZ-VG: RESB-HG: RESB-VG
SC60	CAP\CR08	CAP: CAPA: CAPH: CAPAH
SIP\10P	SIP\10P	
SIP\11P	SIP\11P	
SIP\12P	SIP\12P	
SIP\13P	SIP\13P	
SIP\14P	SIP\14P	
SIP\15P	SIP\15P	
SIP\16P	SIP\16P	
SIP\17P	SIP\17P	
SIP\18P	SIP\18P	
SIP\19P	SIP\19P	

SIP\1P	SIP\1P	
SIP\20P	SIP\20P	
SIP\2P	SIP\2P	
SIP\3P	SIP\3P	
SIP\4P	SIP\4P	
SIP\5P	SIP\5P	
SIP\6P	SIP\6P	
SIP\7P	SIP\7P	
SIP\8P	SIP\8P	
SIP\9P	SIP\9P	
SW-SPDT	PCB-NULL	SW-SPDT
SW-SPST-NC	PCB-NULL	SW-SPST-NC
SW-SPST-NO	PCB-NULL	SW-SPST-NO
TCA5600	DIP18	TCA5600
TCF5600	DIP18	TCA5600
THCT1010	DIP64\900	THCT1010
THCT1010\CC	CC68	THCT1010
THCT2000E	DIP28\600	CC28: CC28A THCT2000E
THCT2000M	DIP28\600	CC28: CC28A THCT2000M
THCT29520	DIP24: DIP24\SOL	29520
THCT29521	DIP24: DIP24\SOL	29521
TIP2955	SOT93	NPN-V: NPN-VA: NPN-H: NPN-HA
TIP3055	SOT93	NPN-V: NPN-VA: NPN-H: NPN-HA
TO-100	TO-100	

\02	DIP14: DIP14\SO	NOR2: NORDM
\02-CC	CC20: CC20A	NOR2: NORDM
\03	DIP14: DIP14\SO	NAND2OC: NANDDMOC
\03-CC	CC20: CC20A	NAND2OC: NANDDMOC
\03A	DIP14: DIP14\SO	NAND2OC: NANDDMOC
\04	DIP14: DIP14\SO	INV: INVL: INVDM: INVDML
\04-CC	CC20: CC20A	INV: INVL: INVDM: INVDML
\04A	DIP14: DIP14\SO	INV: INVL: INVDM: INVDML
\05	DIP14: DIP14\SO	INVOC: INVDMOC
\05-CC	CC20: CC20A	INVOC: INVDMOC
\05A	DIP14: DIP14\SO	INVOC: INVDMOC
\06	DIP14: DIP14\SO	INVOC: INVDMOC
\07	DIP14: DIP14\SO	INVNONOC
\08	DIP14: DIP14\SO	AND2
\08-CC	CC20: CC20A	AND2
\09	DIP14: DIP14\SO	AND2OC
\09-CC	CC20: CC20A	AND2OC
\10	DIP14: DIP14\SO	NAND3
\10-CC	CC20: CC20A	NAND3
\1000	DIP14: DIP14\SO	NAND2: NANDDM
\1000-CC	CC20: CC20A	NAND2: NANDDM
\1000A	DIP14: DIP14\SO	NAND2: NANDDM
\1002-CC	CC20: CC20A	NOR2: NORDM
\1002A	DIP14: DIP14\SO	NOR2: NORDM

\1003-CC	CC20: CC20A	NAND2: NANDDM
\1003A	DIP14: DIP14\SO	NAND2: NANDDM
\1004	DIP14: DIP14\SO	INV: INVL: INVDM: INVDML
\1004-CC	CC20: CC20A	INV: INVL: INVDM: INVDML
\1005	DIP14: DIP14\SO	INV: INVL: INVDM: INVDML
\1005-CC	CC20: CC20A	INV: INVL: INVDM: INVDML
\1008	DIP14: DIP14\SO	AND2
\1008-CC	CC20: CC20A	AND2
\1008A	DIP14: DIP14\SO	AND2
\1010A	DIP14: DIP14\SO	NAND3
\1010A-CC	CC20: CC20A	NAND3
\1011A	DIP14: DIP14\SO	AND3
\1011A-CC	CC20: CC20A	AND3
\1020A	DIP14: DIP14\SO	NAND4
\1020A-CC	CC20: CC20A	NAND4
\1032	DIP14: DIP14\SO	OR2
\1032A	DIP14: DIP14\SO	OR2
\1032A-CC	CC20: CC20A	OR2
\1034	DIP14	INV: INVL: INVDM: INVDML
\1035	DIP14	INVNONOC
\1036	DIP14: DIP14\SO	NOR2: NORDM
\107	DIP14: DIP14\SO	\107
\107-CC	CC20: CC20A	\107
\109	DIP16: DIP16\SO	\109

TO-18	TO-18	
TO-220	TO-220	
TO-220\UP	TO-220\UP	
TO-220\UP\4P	TO-220\UP\4P	
TO-39	TO-39	
TO-5	TO-5	
TO-71	TO-71	
TO-78	TO-78	
TO-8	TO-8	
TO-92	TO-92	
TO-99	TO-99	
TRIAC-TO220	TO-220	TRIAC-V
TRIAC-TO92-KAG	TO-92	TRIAC-V
TRJFETN-TO92	TO-92	JFETN-V
TRJFETP-TO92	TO-92	JFETP-V
TRMFETN-TO220	TO-220	MFETN-V
TRMFETN-TO92	TO-92	MFETN-V
TRMFETP-TO220	TO-220	MFETP-V
TRMFETP-TO92	TO-92	MFETP-V
TRNPN-SOT93	SOT93	NPN-V: NPN-VA: NPN-H: NPN-HA
TRNPN-TO18-EBC	TO-18	NPN-V: NPN-VA: NPN-H: NPN-HA
TRNPN-TO220	TO-220	NPN-V: NPN-VA: NPN-H: NPN-HA
TRNPN-TO3	TO-3	NPN-V: NPN-VA: NPN-H: NPN-HA
TRNPN-TO39	TO-39	NPN-V: NPN-VA: NPN-H: NPN-HA

TRNPN-TO92-BCE	TO-92	NPN-V: NPN-VA: NPN-H: NPN-HA
TRNPN-TO92-BEC	TO-92	NPN-V: NPN-VA: NPN-H: NPN-HA
TRNPN-TO92-CBE	TO-92	NPN-V: NPN-VA: NPN-H: NPN-HA
TRNPN-TO92-EBC	TO-92	NPN-V: NPN-VA: NPN-H: NPN-HA
TRPNP-SOT93	SOT93	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO18-EBC	TO-18	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO220	TO-220	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO3	TO-3	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO39	TO-39	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO92-BCE	TO-92	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO92-BEC	TO-92	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO92-CBE	TO-92	PNP-V: PNP-VA: PNP-H: PNP-HA
TRPNP-TO92-EBC	TO-92	PNP-V: PNP-VA: PNP-H: PNP-HA
TRSCR-TO220	TO-220	SCR-V
TRSCR-TO92-AGK	TO-92	SCR-V
TRSCR-TO92-KGA	TO-92	SCR-V
VRES	VRES	VRESZ-H: VRESZ-V
VRES\TOP\ADJ	VRES\TOP\ADJ	VRESZ-H: VRESZ-V
XTAL1	XTAL1	XTALH
XTAL2	XTAL2	XTALH
\00	DIP14: DIP14\SO	NAND2: NANDDM
\00-CC	CC20: CC20A	NAND2: NANDDM
\00A	DIP14: DIP14\SO	NAND2: NANDDM
\01	DIP14: DIP14\SO	NAND2OC: NANDDMOC

\109-CC	CC20: CC20A	\109
\11	DIP14: DIP14\SO	AND3
\11-CC	CC20: CC20A	AND3
\11000	DIP16: DIP16\SO	NAND2: NANDDM
\11002	DIP16: DIP16\SO	NOR2: NORDM
\11004	DIP20: DIP20\SOL	INV: INVL: INVDM: INVDML
\11008	DIP16: DIP16\SO	AND2
\11010	DIP16: DIP16\SO	NAND3
\11011	DIP16: DIP16\SO	AND3
\11013	DIP14: DIP14\SO	NAND4
\11014	DIP16: DIP16\SO	INVST
\11020	DIP14: DIP14\SO	NAND4
\11021	DIP14: DIP14\SO	AND4
\11027	DIP16: DIP16\SO	NOR3
\11030	DIP14: DIP14\SO	NAND8
\11032	DIP16: DIP16\SO	OR2
\11034	DIP20: DIP20\SOL	INVNON: INVNONL
\11074	DIP14: DIP14\SO	\74
\11086	DIP16: DIP16\SO	XOR
\11109	DIP16: DIP16\SO	\109
\11112	DIP16: DIP16\SO	\112
\11132	DIP16: DIP16\SO	NANDST
\11138	DIP16: DIP16\SO	\138
\11151	DIP16: DIP16\SO	\151

\11158	DIP20:DIP20\SOL	\158
\11160	DIP20:DIP20\SOL	\160
\11162	DIP20:DIP20\SOL	\160
\11175	DIP20:DIP20\SOL	\175
\11190	DIP20:DIP20\SOL	\190
\11191	DIP20:DIP20\SOL	\190
\11194	DIP20:DIP20\SOL	\194
\112	DIP16:DIP16\SO	\112
\112-CC	CC20	\112
\11238	DIP16:DIP16\SO	\238
\11240	DIP24:DIP24\SOL	\240
\11241	DIP24:DIP24\SOL	\244
\11244	DIP24:DIP24\SOL	\244
\11245	DIP24:DIP24\SOL	\573
\11251	DIP16:DIP16\SO	\151
\11253	DIP16:DIP16\SO	\153
\11257	DIP20:DIP20\SOL	\157
\11258	DIP20:DIP20\SOL	\158
\11280	DIP14:DIP14\SO	\280
\112A	DIP16:DIP16\SO	\112
\113	DIP14:DIP14\SO	\113
\113-CC	CC20	\113
\11353	DIP16:DIP16\SO	\352
\11373	DIP24:DIP24\SOL	\573

\11374	DIP24:DIP24\SOL	\574
\113A	DIP14:DIP14\SO	\113
\114	DIP14:DIP14\SO	\114
\114-CC	CC20	\114
\11520	DIP20:DIP20\SOL	\521
\11521	DIP20:DIP20\SOL	\521
\11533	DIP24:DIP24\SOL	\563
\11534	DIP24:DIP24\SOL	\564
\11640	DIP24:DIP24\SOL	\573
\11810	DIP16:DIP16\SO	XNOR
\12	DIP14:DIP14\SO	NAND3
\123	DIP16:DIP16\SO	\221
\1240	DIP20	\240
\1241	DIP20	\244 \244A
\1242	DIP14:DIP14\SO	\1242
\1243	DIP14:DIP14/SO	\1243
\1244	DIP20	\244
\1244A	DIP20:CC20	\244
\1245	DIP20	\573
\125	DIP14:DIP14\SO	\125:\125A
\125-CC	CC20:CC20A	\125:\125A
\126	DIP14:DIP14\SO	\126:\126A
\126-CC	CC20:CC20A	\126:\126A
\128	DIP14:DIP14\SO	NOR2

\13	DIP14: DIP14\SO	NAND4
\131	DIP16	\131
\132	DIP14: DIP14\SO	NANDST
\133	DIP16: DIP16\SO	NAND13
\133-CC	CC20: CC20A	NAND13
\134	DIP16: DIP16\SO	\134
\135	DIP16: DIP16\SO	\135
\136	DIP14: DIP14\SO	XOROC
\137	DIP16: DIP16\SO	\137
\137-CC	CC20: CC20A	\137
\138	DIP16: DIP16\SO	\138: \138S
\138-CC	CC20: CC20A	\138
\139	DIP16: DIP16\SO	\139: \139S
\139-CC	CC20: CC20A	\139
\14	DIP14: DIP14\SO	INVST
\14-CC	CC20: CC20A	INVST
\140	DIP14: DIP14\SO	NAND4
\145	DIP16	\145
\147	DIP16: DIP16\SO	\147
\148	DIP16: DIP16\SO	\148
\15	DIP14: DIP14\SO	AND3
\150	DIP24\600	\150
\151	DIP16: DIP16\SO	\151
\151-CC	CC20: CC20A	\151

\151A	DIP16: DIP16\SO	\151
\152	DIP14	\152
\152A	DIP14	\152
\153	DIP16: DIP16\SO	\153
\153-CC	CC20: CC20A	\153
\154	DIP24\600: DIP24\SOL	\154
\154-CC	CC28: CC28A	\154
\155	DIP16	\155
\156	DIP16	\156
\157	DIP16: DIP16\SO	\157
\157-CC	CC20: CC20A	\157
\158	DIP16: DIP16\SO	\158
\158-CC	CC20: CC20A	\158
\16	DIP14: DIP14\SO	INVOC: INVDMOC
\160	DIP16: DIP16\SO	\160
\160-CC	CC20: CC20A	\160
\160A	DIP16: DIP16\SO	\160
\161	DIP16: DIP16\SO	\160
\161-CC	CC20: CC20A	\160
\161A	DIP16: DIP16\SO	\160
\162	DIP16: DIP16\SO	\160
\162-CC	CC20: CC20A	\160
\1620	DIP20	\1620
\1621	DIP20	\1620

\1622	DIP20	\1620
\1623	DIP20	\1620
\162A	DIP16: DIP16\SO	\160
\163	DIP16: DIP16\SO	\160
\163-CC	CC20: CC20A	\160
\1638	DIP20	\1620
\1639	DIP20	\1620
\163A	DIP16: DIP16\SO	\160
\164	DIP14: DIP14\SO	\164
\164-CC	CC20: CC20A	\164
\1640A	DIP20	\1620
\1641	DIP20	\1620
\1642	DIP20	\1620
\1643	DIP20	\1620
\1644	DIP20	\1620
\1645	DIP20	\1620
\165	DIP16: DIP16\SO	\165
\165-CC	CC20: CC20A	\165
\166	DIP16: DIP16\SO	\166
\166-CC	CC20: CC20A	\166
\168	DIP16: DIP16\SO	\168
\168A	DIP16: DIP16\SO	\168
\169	DIP16: DIP16\SO	\168
\17	DIP14: DIP14\SO	INVNONOC

\170	DIP16: DIP16\SO	\170
\173	DIP16: DIP16\SO	\173
\174	DIP16: DIP16\SO	\174
\174-CC	CC20: CC20A	\174
\175	DIP16: DIP16\SO	\175
\175-CC	CC20: CC20A	\175
\180	DIP14: DIP14\SO	\180
\180-CC	CC20: CC20A	\180
\181	DIP24: DIP24\SOL	\181
\181-CC	CC28: CC28A	\181
\182	DIP16	\182
\189	DIP16	\189
\190	DIP16: DIP16\SO	\190
\190-CC	CC20: CC20A	\190
\191	DIP16: DIP16\SO	\190
\191-CC	CC20: CC20A	\190
\192	DIP16: DIP16\SO	\192
\192-CC	CC20: CC20A	\192
\193	DIP16: DIP16\SO	\192
\193-CC	CC20: CC20A	\192
\194	DIP16: DIP16\SO	\194
\194-CC	CC20: CC20A	\194
\194A	DIP16: DIP16\SO	\194
\195	DIP16: DIP16\SO	\195

\195-CC	CC20: CC20A	\195
\195A	DIP16: DIP16\SO	\195
\198	DIP24: DIP24\SOL	\198
\199	DIP24: DIP24\SOL	\199
\20	DIP14: DIP14\SO	NAND4
\20-CC	CC20: CC20A	NAND4
\21	DIP14: DIP14\SO	AND4
\21-CC	CC20: CC20A	AND4
\219A	DIP16	\219
\22	DIP14	NAND4
\221	DIP16: DIP16\SO	\221
\222	DIP20	\222
\224	DIP16	\224
\225	DIP20	\225
\22A	DIP14: DIP14\SO	NAND4
\230	DIP20	\230
\231	DIP20	\231
\232A	DIP16: DIP16\SO: DIP16\SOL	\232A
\232A-CC	CC20: CC20A	\232A
\233A	DIP20: DIP20\SOL	\233A
\233A-CC	CC20: CC20A	\233A
\237	DIP16: DIP16\SO	\237
\237-CC	CC20: CC20A	\237
\238	DIP16: DIP16\SO	\238

\238-CC	CC20: CC20A	\238
\239	DIP16: DIP16\SO	\239: \239S
\239-CC	CC20: CC20A	\239
\240	DIP20: DIP20\SOL	\240
\240-CC	CC20: CC20A	\240
\240A	DIP20: DIP20\SOL	\240
\241	DIP20: DIP20\SOL	\244: \241
\241-CC	CC20: CC20A	\244: \241
\241A	DIP20: DIP20\SOL	\244: \241
\242	DIP14: DIP14\SO	\242
\242-CC	CC20: CC20A	\242
\242A	DIP14: DIP14\SO	\242
\243	DIP14: DIP14\SO	\242
\243-CC	CC20: CC20A	\242
\243A	DIP14: DIP14\SO	\242
\244	DIP20: DIP20\SOL	\244
\244-CC	CC20: CC20A	\244
\244A	DIP20: DIP20\SOL	\244
\245	DIP20: DIP20\SOL	\573
\245-CC	CC20: CC20A	\573
\245A	DIP20: DIP20\SOL	\573
\247	DIP16	\247
\248	DIP16	\248
\249	DIP16	\248

\250	DIP24\600	\250
\250-CC	CC28: CC28A	\250
\251	DIP16: DIP16\SO	\151
\251-CC	CC20: CC20A	\151
\253	DIP16: DIP16\SO	\153
\253-CC	CC20: CC20A	\153
\257	DIP16: DIP16\SO	\157
\257-CC	CC20: CC20A	\157
\258	DIP16: DIP16\SO	\158
\258-CC	CC20: CC20A	\158
\259	DIP16: DIP16\SO	\259
\259-CC	CC20: CC20A	\259
\26	DIP14: DIP14\SO	NAND2OC: NANDDMOC
\260	DIP14: DIP14\SO	NOR5
\2620	DIP20: DIP20\SOL	\573
\2623	DIP20: DIP20\SOL	\573
\264	DIP16: DIP16\SO	\264
\2640	DIP20: DIP20\SOL	\573
\2645	DIP20: DIP20\SOL	\573
\266	DIP14: DIP14\SO	XNOR
\266-CC	CC20: CC20A	XNOR
\27	DIP14: DIP14\SO	NOR3
\27-CC	CC20: CC20A	NOR3
\273	DIP20: DIP20\SOL	\574

\273-CC	CC20: CC20A	\574
\28	DIP14: DIP14\SO	NOR2: NORDM
\280	DIP14: DIP14\SO	\280
\280-CC	CC20: CC20A	\280
\282	DIP20	\282
\283	DIP16: DIP16\SO	\283
\283-CC	CC20: CC20A	\283
\286	DIP14: DIP14\SO	\286
\286-CC	CC20: CC20A	\286
\28A	DIP14: DIP14\SO	NOR2: NORDM
\290	DIP14: DIP14\SO	\290
\293	DIP14: DIP14\SO	\293
\298	DIP16: DIP16\SO	\298
\298-CC	CC20: CC20A	\298
\299	DIP20: DIP20\SOL	\299
\299-CC	CC20: CC20A	\299
\30	DIP14: DIP14\SO	NAND8
\30-CC	CC20: CC20A	NAND8
\32	DIP14: DIP14\SO	OR2
\32-CC	CC20: CC20A	OR2
\322	DIP20: DIP20\SOL	\322
\322-CC	CC20: CC20A	\322
\323	DIP20: DIP20\SOL	\299
\323-CC	DIP20: DIP20\SOL	\299

\324	DIP14: DIP14\SO	AOPAMPM
\33	DIP14: DIP14\SO	NOR2OC: NORDMOC
\33A	DIP14: DIP14\SO	NOR2OC: NORDMOC
\34	DIP14: DIP14\SO	INVNON: INVNONL
\35	DIP14: DIP14\SO	INVNON: INVNONL
\352	DIP16: DIP16\SO	\352
\352-CC	CC20: CC20A	\352
\353	DIP16: DIP16\SO	\352
\353-CC	CC20: CC20A	\352
\354	DIP20: DIP20\SOL	\354
\354-CC	CC20: CC20A	\354
\356	DIP20: DIP20\SOL	\356
\356-CC	CC20: CC20A	\356
\363	DIP20: DIP20\SOL	\573
\364	DIP20: DIP20\SOL	\574
\365	DIP16: DIP16\SO	\365
\365-CC	CC20: CC20A	\365
\366	DIP16: DIP16\SO	\366
\366-CC	CC20: CC20A	\366
\367	DIP16: DIP16\SO	\367A: \367B
\367-CC	CC20: CC20A	\367A: \367B
\368	DIP16: DIP16\SO	\368A: \368B
\368-CC	CC20: CC20A	\368A: \368B
\37	DIP14: DIP14\SO	NAND2: NANDM

\373	DIP20: DIP20\SOL	\573
\373-CC	CC20: CC20A	\573
\374	DIP20: DIP20\SOL	\574
\374-CC	CC20: CC20A	\574
\375	DIP16: DIP16\SO	\75
\377	DIP20: DIP20\SOL	\574
\377-CC	CC20: CC20A	\574
\378	DIP16: DIP16\SO	\174
\379	DIP16: DIP16\SO	\175
\37A	DIP14: DIP14\SO	NAND2: NANDDM
\38	DIP14: DIP14\SO	NAND2OC: NANDDMOC
\381	DIP20	\381
\386	DIP14: DIP14\SO	XOR
\386-CC	CC20: CC20A	XOR
\38A	DIP14: DIP14\SO	NAND2OC: NANDDMOC
\39	DIP14: DIP14\SO	NAND2OC: NANDDMOC
\390	DIP16: DIP16\SO	\390
\390-CC	CC20: CC20A	\390
\393	DIP14: DIP14\SO	\393
\393-CC	CC20: CC20A	\390
\395	DIP16: DIP16\SO	\395
\395-CC	CC20: CC20A	\395
\395A	DIP16: DIP16\SO	\395
\40	DIP14: DIP14\SO	NAND4

\4002	DIP14: DIP14\SO	NOR4
\4016	DIP14: DIP14\SO	4066: 4066V
\4040	DIP16: DIP16\SO	\4040
\4046	DIP16: DIP16\SO	\4046
\4049	DIP16: DIP16\SO	INV: INVL: INVDM: INVDML
\4051	DIP16: DIP16\SO	4051
\4052	DIP16: DIP16\SO	4052
\4053	DIP16: DIP16\SO	\4053
\4060	DIP16: DIP16\SO	\4060
\4061	DIP16: DIP16\SO	4060
\4066	DIP14: DIP14\SO	4066: 4066V
\4075	DIP14: DIP14\SO	OR3
\4094	DIP16: DIP16\SO	4094
\40A	DIP14: DIP14\SO	NAND4
\412	DIP24: DIP24\SOL: DIP24\300	\412
\42	DIP16: DIP16\SO	\42
\42A	DIP16: DIP16\SO	\42
\4316	DIP16: DIP16\SO	\4316
\432	DIP24: DIP24\SOL	\432
\4351	DIP20: DIP20\SOL	4351
\4352	DIP20: DIP20\SOL	4352
\4353	DIP20: DIP20\SOL	4353
\4511	DIP16: DIP16\SO	4511
\4518	DIP16: DIP16\SO	\4518

\4520	DIP16: DIP16\SO	\4518
\4538	DIP16: DIP16\SO	\4538
\465A	DIP20: DIP20\SOL	\541
\466A	DIP20: DIP20\SOL	\540
\467A	DIP20: DIP20\SOL	\244
\468A	DIP20: DIP20\SOL	\240
\47	DIP16	\47
\470	DIP20	\470
\471	DIP20	\470
\472	DIP20	\472
\4724	DIP16: DIP16\SO	4724
\474	DIP24\600	\474
\475	DIP24\600	\475
\47A	DIP16	\47
\49	DIP14	\49
\490	DIP16: DIP16\SO	\490
\490-CC	CC20: CC20A	\390
\51	DIP14: DIP14\SO	\51A: \51B
\51-CC	CC20: CC20A	\51A: \51B
\518	DIP20	\518
\519	DIP20	\518
\520	DIP20	\520
\521	DIP20	\520
\522	DIP20	\520

\526	DIP20	\526
\527	DIP20	\527
\528	DIP16	\528
\533	DIP20: DIP20\SOL	\563
\533-CC	CC20: CC20A	\563
\534	DIP20: DIP20\SOL	\564
\534-CC	CC20: CC20A	\564
\538	DIP20: DIP20\SOL	\538
\539	DIP20: DIP20\SOL	\539
\540	DIP20: DIP20\SOL	\540
\540-CC	CC20: CC20A	\540
\541	DIP20: DIP20\SOL	\541
\541-CC	CC20: CC20A	\541
\560A	DIP20	\560A
\561A	DIP20	\560A
\563	DIP20: DIP20\SOL	\563
\563-CC	CC20: CC20A	\563
\564	DIP20: DIP20\SOL	\564
\564-CC	CC20: CC20A	\564
\568	DIP20: DIP20\SOL	\568
\568A	DIP20: DIP20\SOL	\568
\569	DIP20: DIP20\SOL	\568
\569A	DIP20: DIP20\SOL	\568
\573	DIP20: DIP20\SOL	\573

\573-CC	CC20: CC20A	\573
\574	DIP20: DIP20\SOL	\574
\574-CC	CC20: CC20A	\574
\575	DIP24	\575
\576	DIP20	\576
\577	DIP24	\577
\580	DIP20	\580
\589	DIP16	\589
\590	DIP16: DIP16\SO	\590
\590-CC	CC20: CC20A	\590
\594	DIP16: DIP16\SO	\594
\594-CC	CC20: CC20A	\590
\595	DIP16: DIP16\SO	\594
\595-CC	CC20: CC20A	\590
\604	DIP28\600	\604
\604-CC	CC28: CC28A	\604
\610	DIP40\600	\610
\610-CC	CC44: CC44A	\610
\611	DIP40\600	\610
\611-CC	CC44: CC44A	\610
\612	DIP40\600	\610
\612-CC	CC44: CC44A	\610
\613	DIP40\600	\610
\613-CC	CC44: CC44A	\610

\616	DIP40\600	\616
\616-CC	CC28: CC28A	\616
\617	DIP40\600	\616
\617-CC	CC28: CC28A	\616
\620	DIP20: DIP20\SOL	\573
\620-CC	CC20: CC20A	\573
\620A	DIP20: DIP20\SOL	\573
\621	DIP20: DIP20\SOL	\573
\621A	DIP20: DIP20\SOL	\573
\622	DIP20: DIP20\SOL	\573
\622A	DIP20: DIP20\SOL	\573
\623	DIP20: DIP20\SOL	\573
\623-CC	CC20: CC20A	\573
\623A	DIP20: DIP20\SOL	\573
\6301	DIP52\600	\6301
\6302	DIP52\600	\6302
\632	DIP52\600	\632
\632-CC	CC68: CC68A	\632
\632A	DIP52\600	\632
\632A-CC	CC68: CC68A	\632
\633	DIP52\600	\632
\633-CC	CC68: CC68A	\632
\634	DIP48\600	\634
\634-CC	CC68: CC68A	\634

\635	DIP48\600	\634
\635-CC	CC68: CC68A	\634
\638	DIP20: DIP20\SOL	\573
\638A	DIP20: DIP20\SOL	\573
\639	DIP20: DIP20\SOL	\573
\639A	DIP20: DIP20\SOL	\573
\64	DIP14	\64
\640	DIP20: DIP20\SOL	\573
\640-CC	CC20: CC20A	\573
\640A	DIP20: DIP20\SOL	\573
\641	DIP20: DIP20\SOL	\573
\641A	DIP20: DIP20\SOL	\573
\642	DIP20: DIP20\SOL	\573
\642A	DIP20: DIP20\SOL	\573
\643	DIP20: DIP20\SOL	\573
\643-CC	CC20: CC20A	\573
\643A	DIP20: DIP20\SOL	\573
\644	DIP20: DIP20\SOL	\573
\644A	DIP20: DIP20\SOL	\573
\645	DIP20: DIP20\SOL	\573
\645-CC	CC20: CC20A	\573
\645A	DIP20: DIP20\SOL	\573
\646	DIP24: DIP24\SOL	\646
\646-CC	CC28: CC28A	\646

\647	DIP24: DIP24\SOL	\646
\647-CC	CC28: CC28A	\646
\648	DIP24: DIP24\SOL	\646
\648-CC	CC28: CC28A	\646
\649	DIP24: DIP24\SOL	\646
\649-CC	CC28: CC28A	\646
\651	DIP24: DIP24\SOL: DIP24\300	\646
\651-CC	CC28: CC28A	\646
\652	DIP24: DIP24\SOL: DIP24\300	\646
\652-CC	CC28: CC28A	\646
\653	DIP24: DIP24\SOL: DIP24\300	\646
\653-CC	CC28: CC28A	\646
\654	DIP24: DIP24\SOL: DIP24\300	\646
\654-CC	CC28: CC28A	\646
\658	DIP24: DIP24\SOL	\658
\658-CC	CC28: CC28A	\658
\659	DIP24: DIP24\SOL	\658
\659-CC	CC28: CC28A	\658
\664	DIP24: DIP24\SOL	\658
\664-CC	CC28: CC28A	\658
\665	DIP24: DIP24\SOL	\658
\665-CC	CC28: CC28A	\658
\666	DIP24: DIP24\300: DIP24\SOL	\666
\666-CC	CC28: CC28A	\666

\667	DIP24: DIP24\SOL: DIP24\300	\667
\667-CC	CC28: CC28A	\667
\670	DIP16	\670
\673	DIP24: DIP24\SOL	\673
\674	DIP24: DIP24\SOL	\674
\675	DIP24: DIP24\SOL	\675
\676	DIP24: DIP24\SOL	\674
\677	DIP24: DIP24\SOL	\677
\677-CC	CC28: CC28A	\677
\678	DIP24: DIP24\SOL	\678
\678-CC	CC28: CC28A	\678
\679	DIP20: DIP20\SOL	\679
\679-CC	CC20: CC20A	\679
\680	DIP20: DIP20\SOL	\680
\680-CC	CC20: CC20A	\679
\682	DIP20: DIP20\SOL	\682
\682-CC	CC20: CC20A	\682
\684	DIP20: DIP20\SOL	\682
\684-CC	CC20: CC20A	\682
\686	DIP24\300: DIP24\SOL	\686
\686-CC	CC28: CC28A	\686
\688	DIP20: DIP20\SOL	\521
\688-CC	CC20: CC20A	\521
\689	DIP20: DIP20\SOL	\521

\690	DIP20: DIP20\SOL	\690
\690-CC	CC20: CC20A	\690
\691	DIP20: DIP20\SOL	\690
\691-CC	CC20: CC20A	\690
\692	DIP20: DIP20\SOL	\690
\693	DIP20: DIP20\SOL	\690
\693-CC	CC20: CC20A	\690
\696	DIP20: DIP20\SOL	\696
\696-CC	CC20: CC20A	\696
\697	DIP20: DIP20\SOL	\696
\697-CC	CC20: CC20A	\696
\698	DIP20: DIP20\SOL	\696
\698-CC	CC20: CC20A	\696
\699	DIP20: DIP20\SOL	\696
\699-CC	CC20: CC20A	\696
\70	DIP14	\70
\7022	DIP16: DIP16\SO	\7022
\7046	DIP16: DIP16\SO	4046
\7074	DIP24	\7074
\7074-CC	CC28: CC28A	\7074
\72	DIP14	\72
\7266	DIP14: DIP14\SO	XNOR
\7266-CC	CC20: CC20A	XNOR
\73	DIP14: DIP14\SO	\73

\74	DIP14: DIP14\SO	\74
\74-CC	CC20: CC20A	\74
\75	DIP16: DIP16\SO	\75
\756	DIP20	\756
\758	DIP14	\758
\759	DIP14	\758
\76	DIP16: DIP16\SO	\76
\760	DIP20: DIP20\SOL	\244
\762	DIP20	\762: \762A
\763	DIP20	\763: \763A
\77	DIP14	\77
\78	DIP14: DIP14\SO	\78
\800	DIP20	NAND4
\8003	DIP8	NAND2: NANDDM
\802	DIP20	NOR4
\804	DIP20	NAND2: NANDDM
\804-CC	CC20: CC20A	NAND2: NANDDM
\804A	DIP20	NAND2: NANDDM
\805	DIP20	NOR2: NORDM
\805-CC	CC20: CC20A	NOR2: NORDM
\805A	DIP20	NOR2: NORDM
\808	DIP20	AND2
\808-CC	CC20: CC20A	AND2
\808A	DIP20	AND2

\821	DIP24:DIP24\SOL:DIP24\300	821
\821-CC	CC28:CC28A	\821
\822	DIP24:DIP24\SOL:DIP24\300	\822
\822-CC	CC28:CC28A	\822
\823	DIP24:DIP24\SOL:DIP24\300	\823
\823-CC	CC28:CC28A	\823
\824	DIP24:DIP24\SOL:DIP24\300	\824
\824-CC	CC28:CC28A	\824
\825	DIP24:DIP24\SOL:DIP24\300	\825
\825-CC	CC28:CC28A	\825
\826	DIP24:DIP24\SOL:DIP24\300	\826
\826-CC	CC28:CC28A	\826
\832	DIP20	OR2
\832-CC	CC20:CC20A	OR2
\832A	DIP20	OR2
\83A	DIP16:DIP16\SO	\83A
\841	DIP24:DIP24\SOL:DIP24\300	\841
\841-CC	CC28:CC28A	\841
\842	DIP24:DIP24\SOL:DIP24\300	\842
\842-CC	CC28:CC28A	\842
\843	DIP24:DIP24\SOL:DIP24\300	\843
\844	DIP24:DIP24\SOL:DIP24\300	\843
\845	DIP24:DIP24\SOL:DIP24\300	\845
\845-CC	CC28:CC28A	\845

\846	DIP24:DIP24\SOL:DIP24\300	\846
\846-CC	CC28:CC28A	\846
\85	DIP16:DIP16\SO	\85
\85-CC	CC20:CC20A	\85
\850	DIP28\600	\850
\850-CC	CC28:CC28A	\850
\851	DIP28\600	\851
\851-CC	CC28:CC28A	\851
\852	DIP24	\852
\852-CC	CC28:CC28A	\852
\856	DIP24	\856
\856-CC	CC28:CC28A	\856
\857	DIP24	\857
\86	DIP14:DIP14\SO	XOR
\86-CC	CC20:CC20A	XOR
\866	DIP28\600	\866
\867	DIP24	\867
\867-CC	CC28:CC28A	\867
\869	DIP24	\867
\869-CC	CC28:CC28A	\867
\870	DIP24	\870
\870-CC	CC28:CC28A	\870
\871	DIP28\600	\871
\871-CC	CC28:CC28A	\871

\873	DIP24	\873
\874	DIP24	\874
\876	DIP24	\876
\877	DIP24	\877
\877-CC	CC28: CC28A	\877
\878	DIP24	\874
\879	DIP24	\876
\880	DIP24	\880
\881	DIP24: DIP24\SOL	\181
\881-CC	CC28: CC28A	\181
\881A	DIP24: DIP24\SOL	\181
\881A-CC	CC28: CC28A	\181
\882	DIP24: DIP24\SOL: DIP24\300	\882
\882A	DIP24: DIP24\SOL: DIP24\300	\882
\882A-CC	CC28: CC28A	\882
\885	DIP24	\885
\885-CC	CC28: CC28A	\885
\89	DIP16	\89
\90	DIP14	\90
\90A	DIP14	\90
\91	DIP14: DIP14\SO	\91
\91A	DIP14: DIP14\SO	\91
\92	DIP14: DIP14\SO	\92
\92A	DIP14: DIP14\SO	\92

\93	DIP14:DIP14\SO	\92
\94	DIP16	\94
\95	DIP14:DIP14\SO	\95
\95A	DIP14:DIP14\SO	\95
\96	DIP16	\96
\963	DIP20:DIP20\SOL	\963
\963-CC	CC20:CC20A	\963
\964	DIP20:DIP20\SOL	\964
\964-CC	DIP20:DIP20\SOL	\964
\97	DIP16	\97
\990	DIP20:DIP20\SOL	\990
\990-CC	CC20:CC20A	\990
\991	DIP20:DIP20\SOL	\991
\991-CC	CC20:CC20A	\991
\992	DIP24:DIP24\SOL	\992
\993	DIP24:DIP24\SOL	\993
\993-CC	CC28:CC28A	\993 \993-DW DIP24\SOL
\994	DIP24:DIP24\SOL	\994
\994-CC	CC28:CC28A	\994
\995	DIP24:DIP24\SOL	\995
\996	DIP24:DIP24\SOL	\996
\996-CC	CC28:CC28A	\996

APPENDIX B

PADS™-LOGIC

DECAL PLOTS

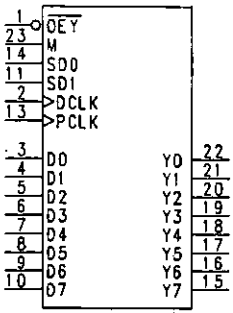
APPENDIX B

DECAL PLOTS

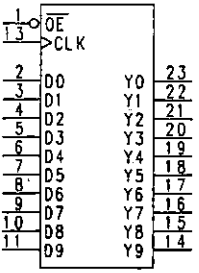
This Appendix contains some 113 Drawing Sheets showing plots of Schematic Decals. Each drawing sheet contains a number of Schematic Decal Plots. An Index of these Decals is as follows:

INDEX

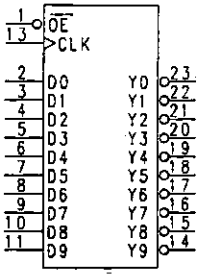
DECALS	PAGE NO.
AM29818 THRU AM29864	1,2
CAPS	3
DIPS	4 - 8
GATES	9 - 11
GENERIC DIPS	12 - 16
MISC. COMPONENTS	82 - 83
OP AMPS	~~17~~ 19
PALS	18 - 23
PINS	26
POWER DEVICES	27
R* (PROMS/RAMS)	28 - 33
RELAYS	34
RESISTORS	35
TXXXXXX	36
1XXXXXX	37 - 40
2XXXXXX	41 - 47
3XXXXXX	48 - 52
4XXXXXX	53 - 57
6XXXXXX	58 - 63
7XXXXXX	64
8XXXXXX	65 - 81
9XXXXXX	82 - 84
TTL 54\74 SERIES \107 - \996	85 - 113



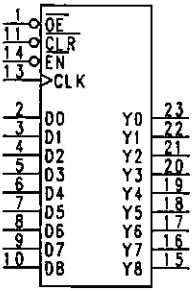
29818



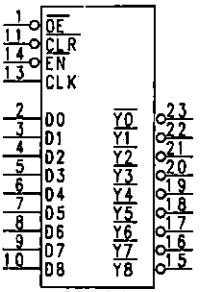
29821



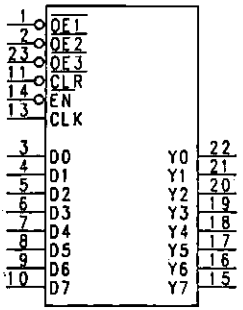
29822



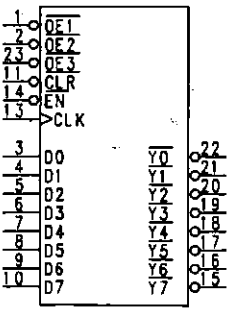
29823



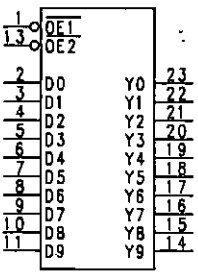
29824



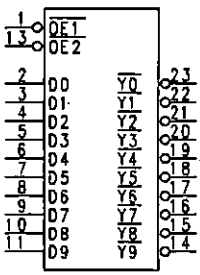
29825



29826

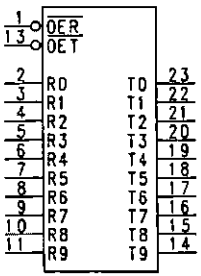


29827

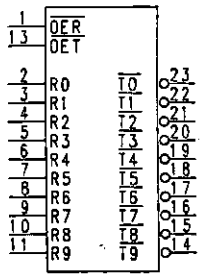


29828

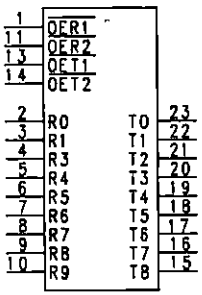
DRAWN BY: S.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE AM298**		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 1 OF 113



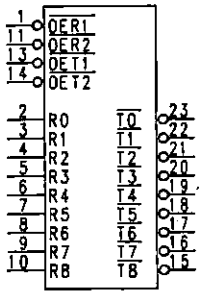
29861



29862

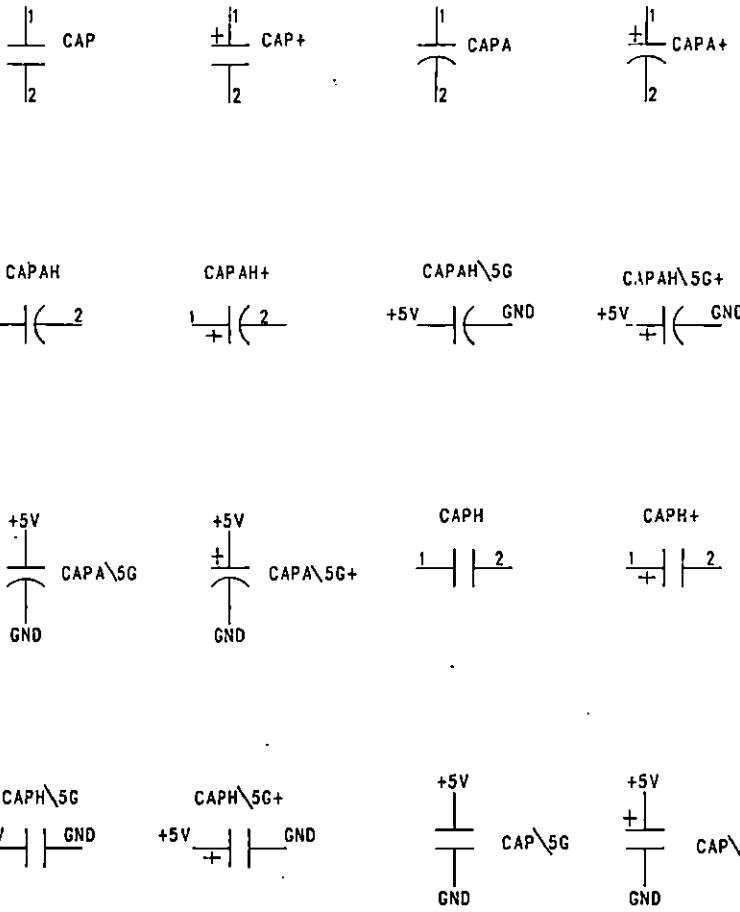


29863

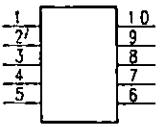


29864

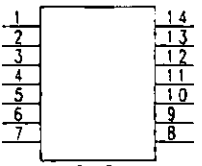
DRAWN BY: fp	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: TP	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE AM298**		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 2 OF 113



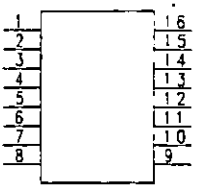
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE CAPS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 3 OF 113



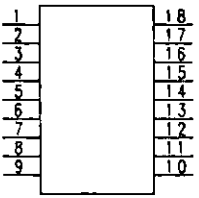
DIP10



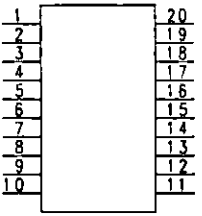
DIP14



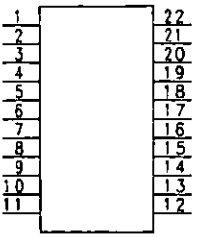
DIP16



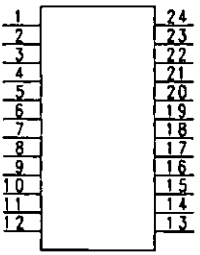
DIP18



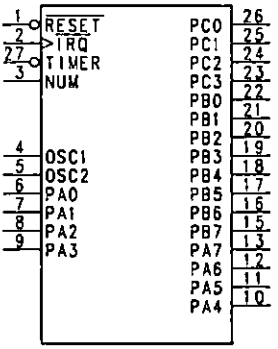
DIP20



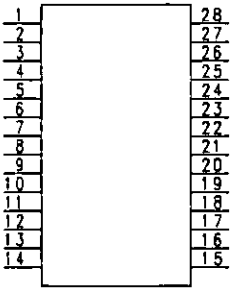
DIP22



DIP24

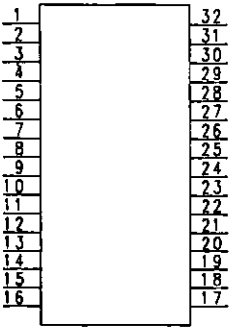


DIP26

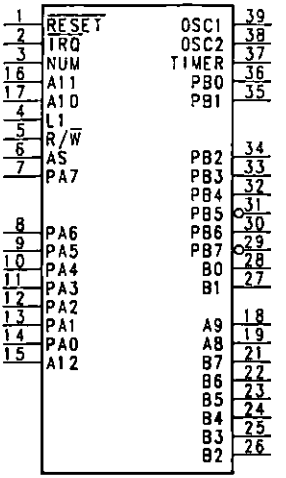


DIP28

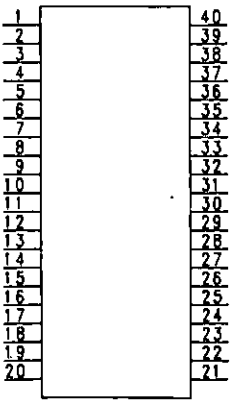
DRAWN BY: J.L.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: RD	DATE 1-3-90		
APPROVED BY: [illegible]	DATE 1-3-90	TITLE DIP	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 4 OF 113



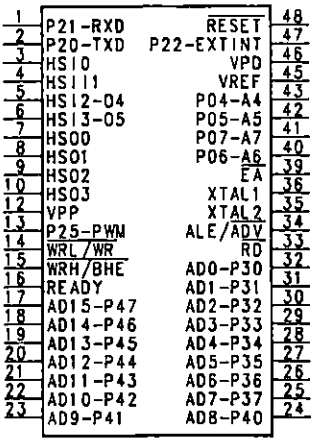
DIP32



DIP38

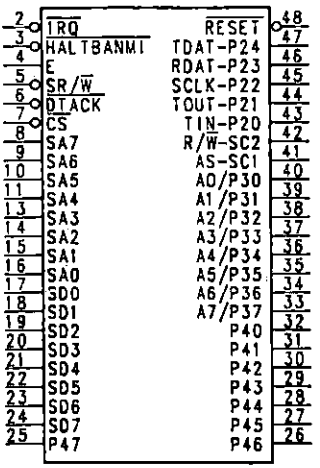


DIP40

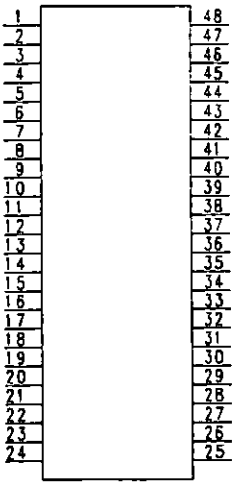


DIP44

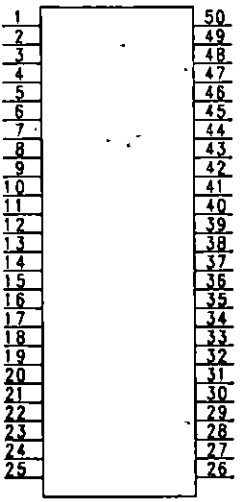
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: @	DATE 1-3-90		
APPROVED BY:	DATE 1-3-90	TITLE DIP	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 5 OF 113



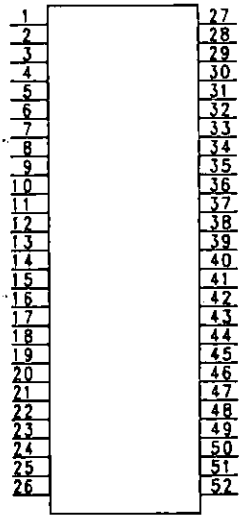
DIP46



DIP48

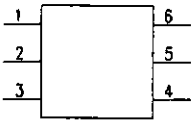


DIP50

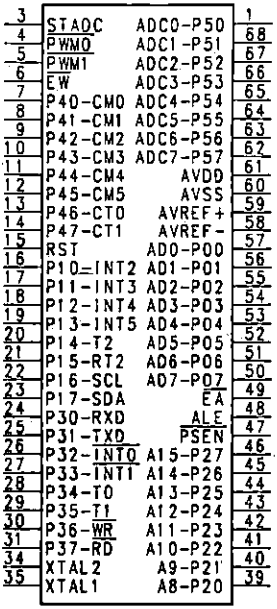


DIP52

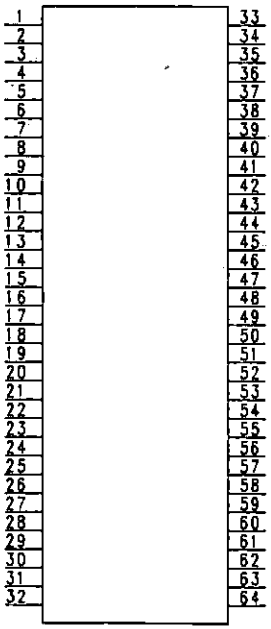
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: (illegible)	DATE 1-3-90			
APPROVED BY: (illegible)	DATE 1-3-90	TITLE DIP		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 6 OF 113



DIP6

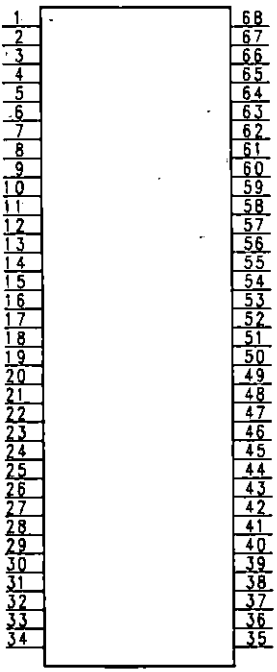


DIP62

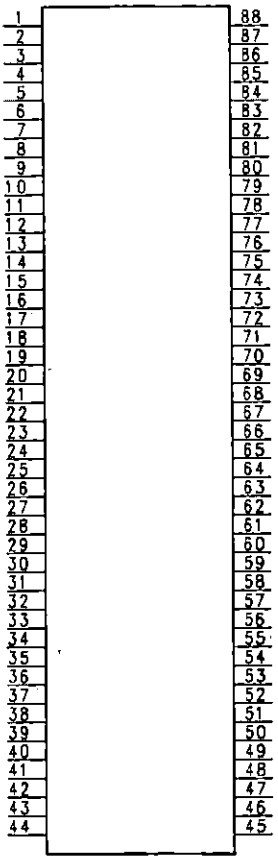


DIP64

DRAWN BY: J. P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY:	DATE 1-3-90		
APPROVED BY:	DATE 1-3-90	TITLE DIP	
REVISION:	DATE		
JOB NAME:		REV. A	SCALE 1:1	SHEET 7 OF 113

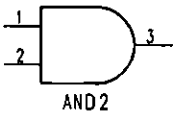


DIP8

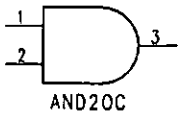


DIP88

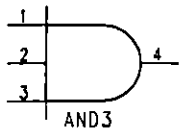
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE	TITLE DIP		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 8 OF 113



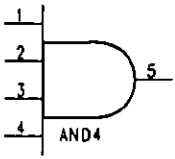
AND2



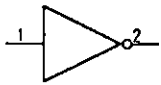
AND2OC



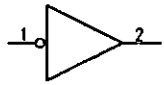
AND3



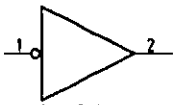
AND4



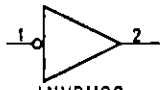
INV



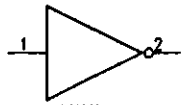
INVDM



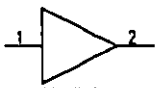
INVDML



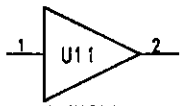
INVDMOC



INVL

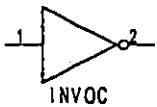


INVNON

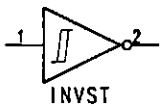


INVNONL

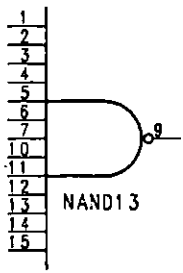
INVNONOC



INVOC



INVST



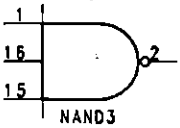
NAND13

DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE GATES		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 9 OF 113

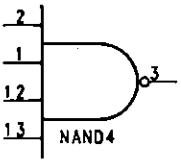
NAND2

NAND2DM

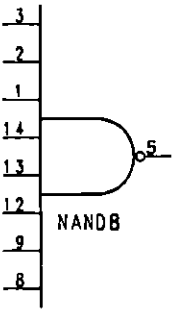
NAND2OC



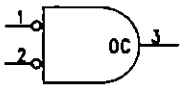
NAND3



NAND4



NAND8

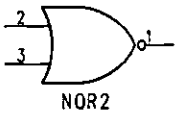


NANDDMOC

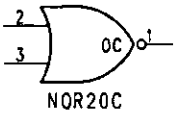
NANDDM

NANDST

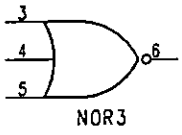
DRAWN BY: J. P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: Chas	DATE 1-3-90	TITLE GATES		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 10 OF 113



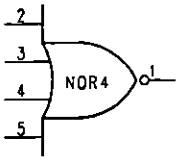
NOR2

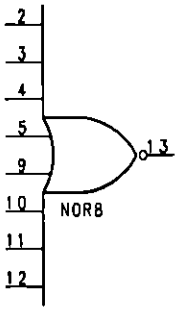


NOR2OC

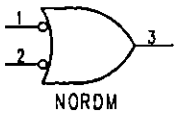


NOR3

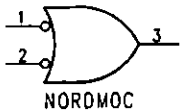




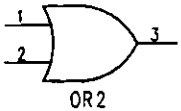
NOR8



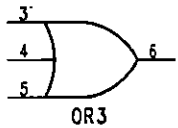
NORDM



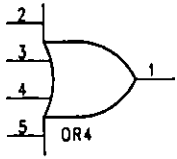
NORDMOC



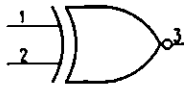
OR2



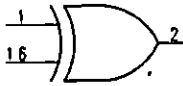
OR3



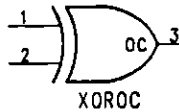
OR4



XNOR

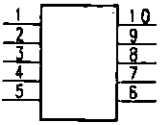


XOR

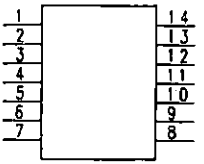


XOROC

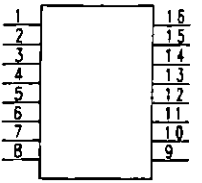
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE GATES		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 11 OF 113



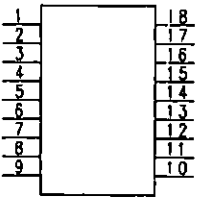
GEN\DIP10



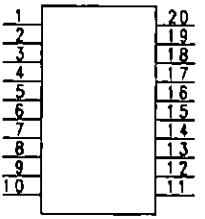
GEN\DIP14



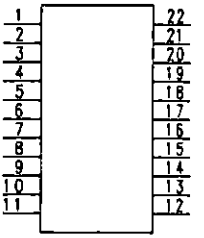
GEN\DIP16



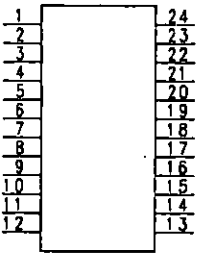
GEN\DIP18



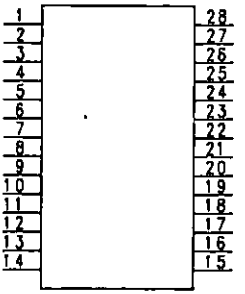
GEN\DIP20



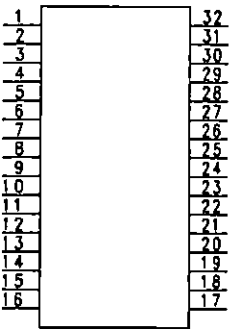
GEN\DIP22



GEN\DIP24

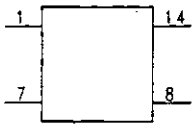


GEN\DIP28

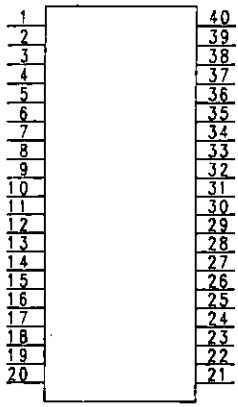


GEN\DIP32

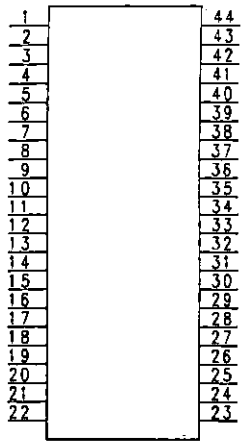
DRAWN BY: fp	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE GENERIC DIPS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 12 OF 113



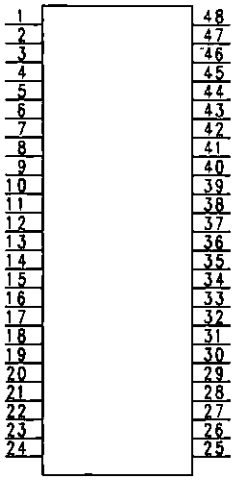
GEN\DIP4(14)



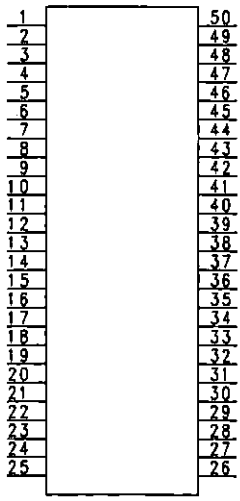
GEN\DIP40



GEN\DIP44

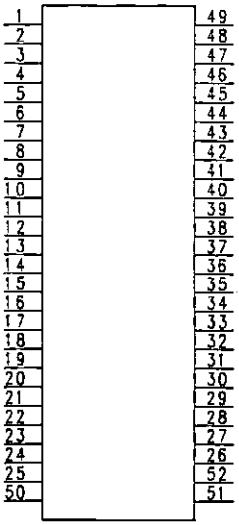


GEN\DIP48

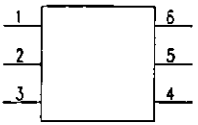


GEN\DIP50

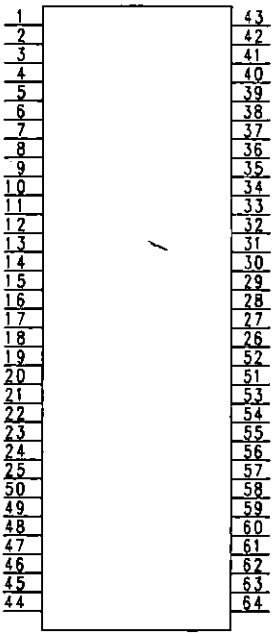
DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE GENERIC DIPS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 13 OF 113



GEN\DIP52

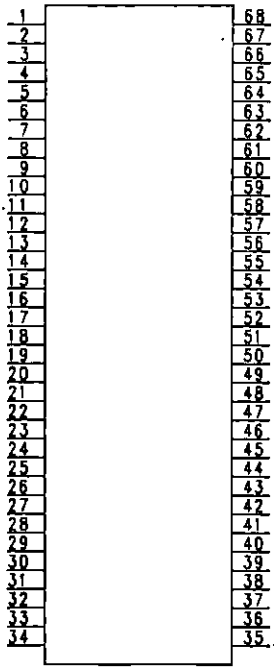


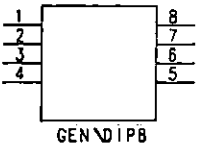
GEN\DIP6



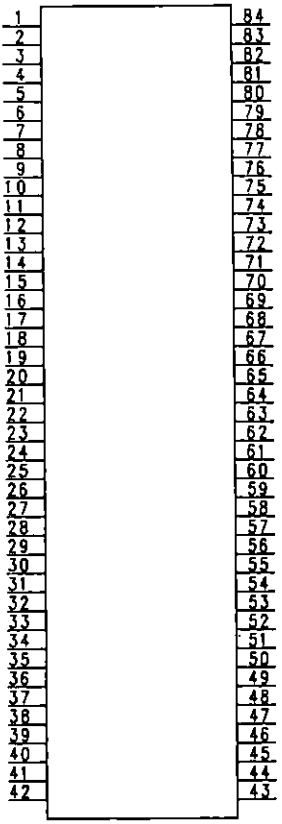
GEN\DIP64

DRAWN BY: J. P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE GENERIC DIPS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 14 OF 113





DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: TC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE GENERIC DIPS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 15 OF 113

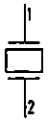


GEN\DIP84

DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: (illegible)	DATE 1-3-90			
APPROVED BY: (illegible)	DATE 1-3-90	TITLE GENERIC DIPS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 16 OF 113

DIO-H

DIO-V

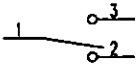


XTAL

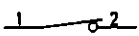
XTALH

EXTIN

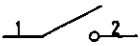
EXTOUT



SW-SPDT



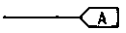
SW-SPST-NC



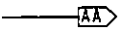
SW-SPST-NO

DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE DIODE, XTAL, IN/OUT AND SWITCHES		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 17 OF 113

OFF PAGE SYMBOLS



REFIN

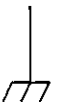


REFOUT

GROUND SYMBOLS

GND

AGND



CHGND

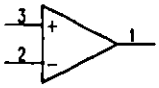
VOLTAGE SYMBOLS

+5V

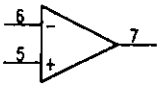
BUBBLE

Y

DRAWN BY: L. P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: DC	DATE 1-3-90		
APPROVED BY: [illegible]	DATE 1-3-90	TITLE OFFPAGE, POWER, AND GROUND	
REVISION:	DATE		
JOB NAME:		REV. A	SCALE 1:1	SHEET 18 OF 113

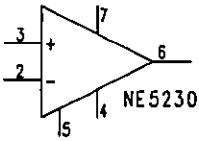


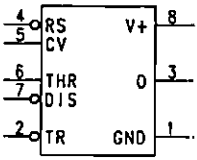
AOPAMP



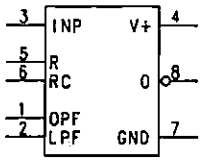
AOPAMPM

AOPAMPPN



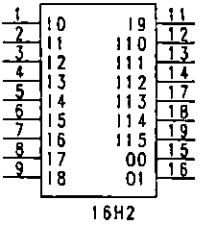


NE555

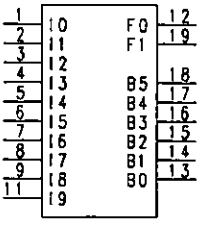


NE567

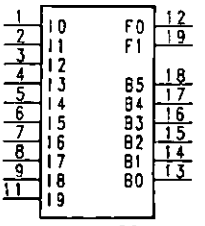
DRAWN BY: J. P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE OP AMPS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 19 OF 113



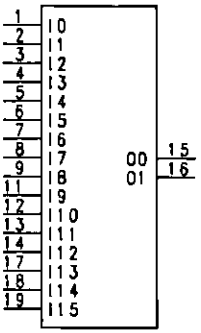
16H2



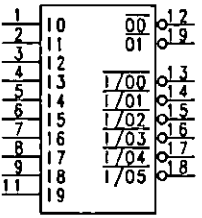
16H8



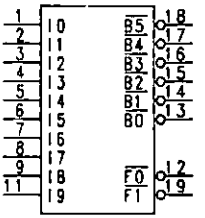
16HD8



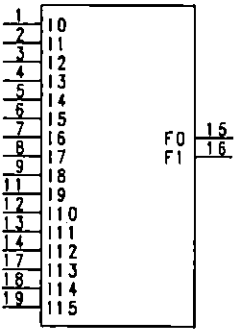
16L2



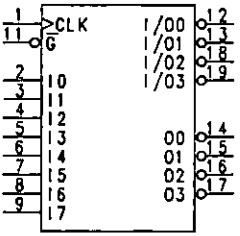
16L8



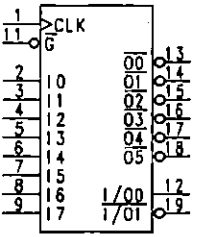
16LD8



16P2

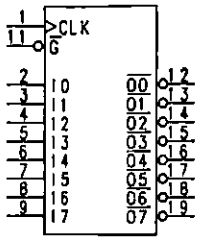


16R4

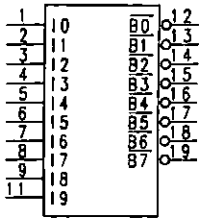


16R6

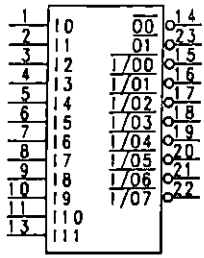
DRAWN BY: L.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: [illegible]	DATE 1-3-90		
APPROVED BY: [illegible]	DATE 1-3-90	TITLE PALS	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 20 OF 113



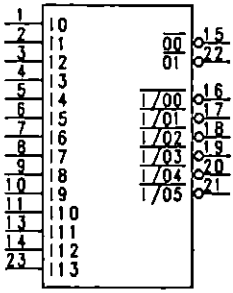
16R8



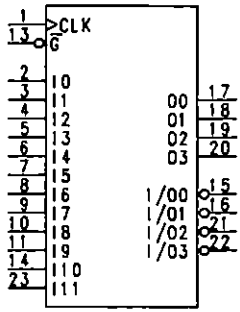
18P8



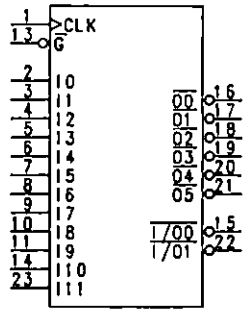
20L10



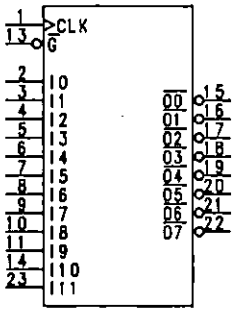
20L8



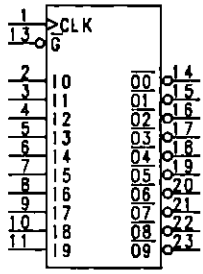
20R4



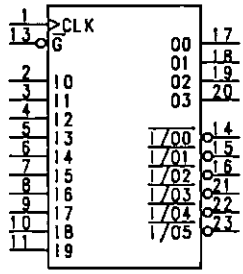
20R6



20R8

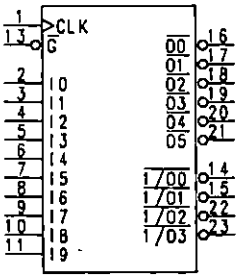


20RP10

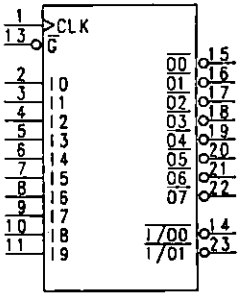


20RP4

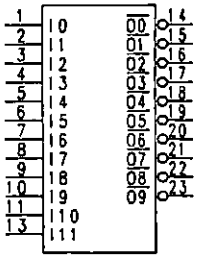
DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE PALS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 21 OF 113



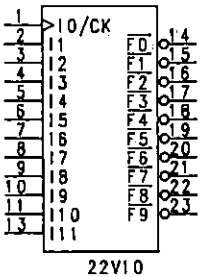
20RP6



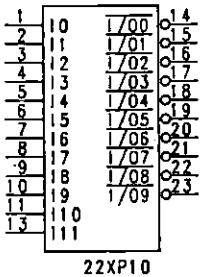
20RP8



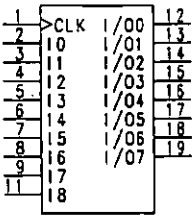
22P10



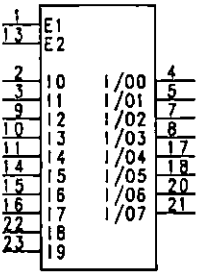
22V10



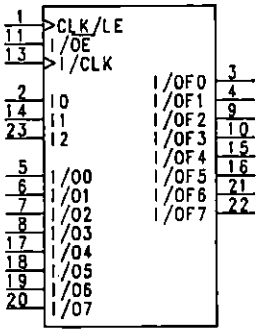
22XP10



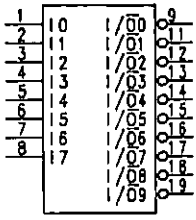
23S8



10H20

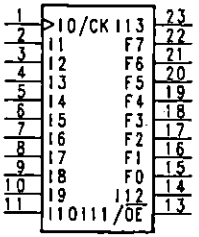


29M16

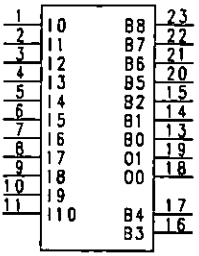


PLS153

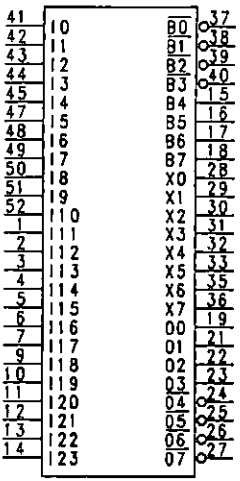
DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE PALS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 22 OF 113



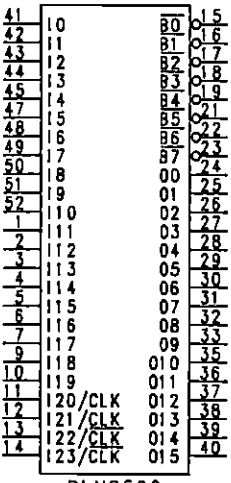
20V8



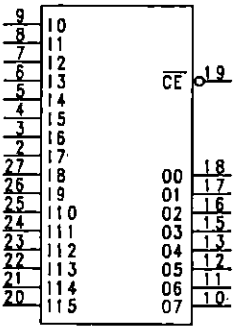
PLC473



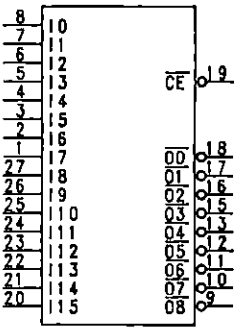
PLHS501



PLHS502

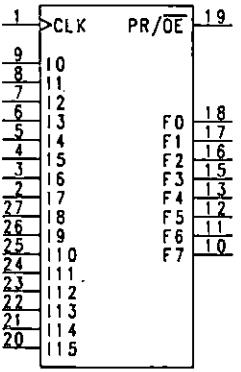


PLS100

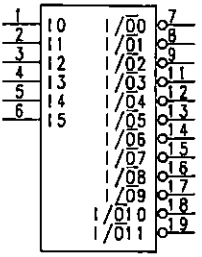


PLS103

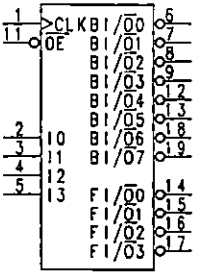
DRAWN BY: P.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE PALS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 23 OF 113



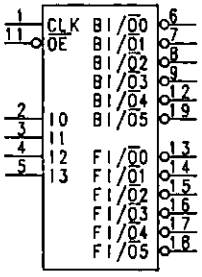
PLS105



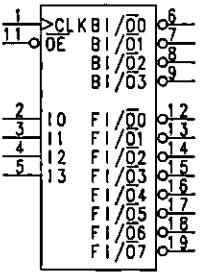
PLS151



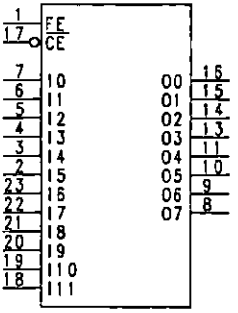
PLS155



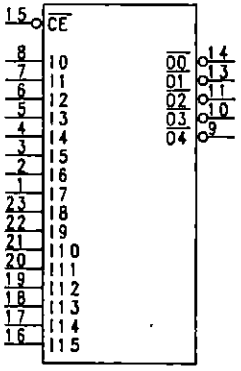
PLS157



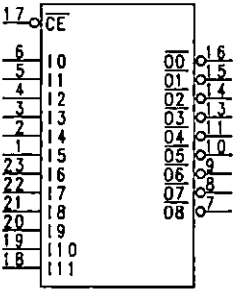
PLS159



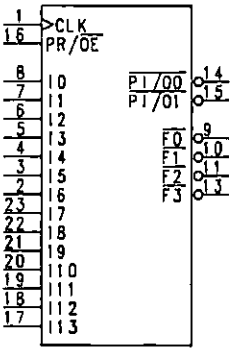
PLS161



PLS162

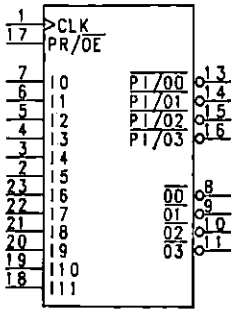


PLS163

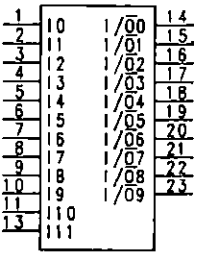


PLS167

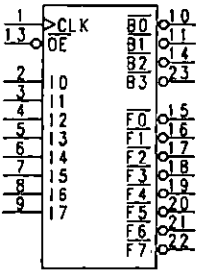
DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY:	DATE 1-3-90		
APPROVED BY:	DATE 1-3-90	TITLE PALS	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 24 OF 113



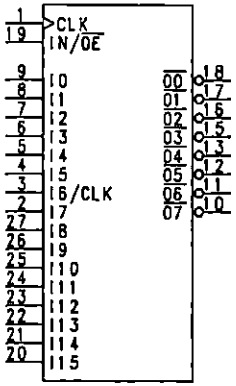
PLS168



PLS173



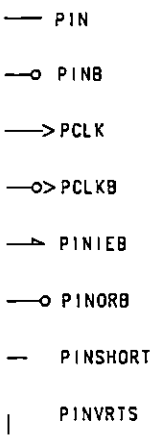
PLS179



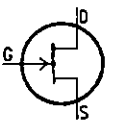
PLUS405

DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DP	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE PALS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 25 OF 113

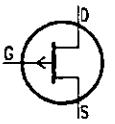
PINS



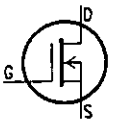
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DO	DATE 1-3-90			
APPROVED BY:	DATE	TITLE PINS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 26 OF 113



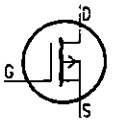
JFETN-V



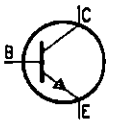
JFETP-V



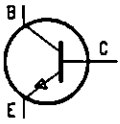
MFETN-V



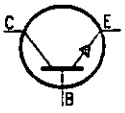
MFETP-V



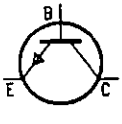
NPN-V



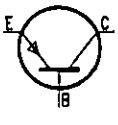
NPN-VA



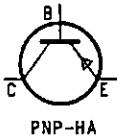
NPN-H



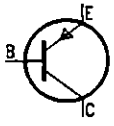
NPN-HA



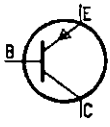
PNP-H



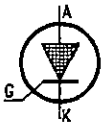
PNP-HA



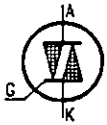
PNP-V



PNP-VA

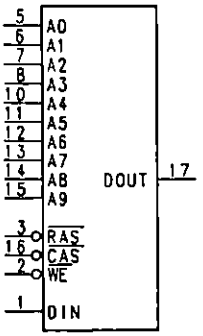


SCR-V

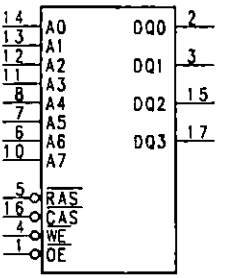


TRIAC-V

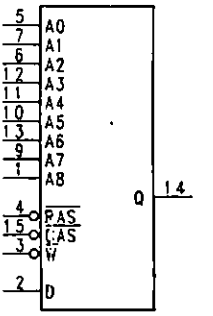
DRAWN BY: J P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: RD.	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE POWER DEVICES		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 27 OF 113



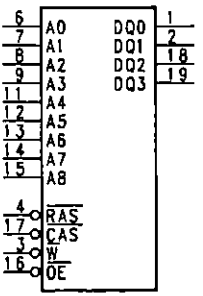
RD1024K1



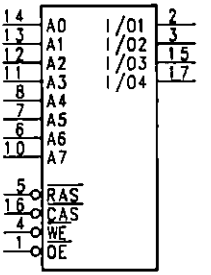
RD16K4



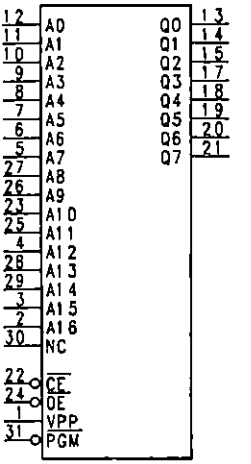
RD256K1



RD256K4

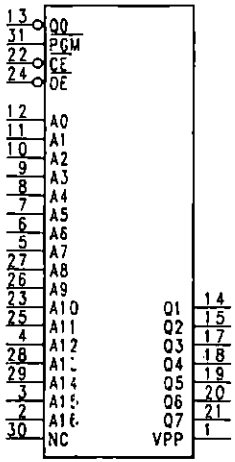


RD64K4

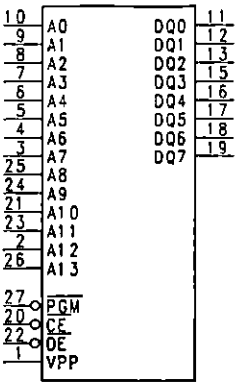


RE128K8

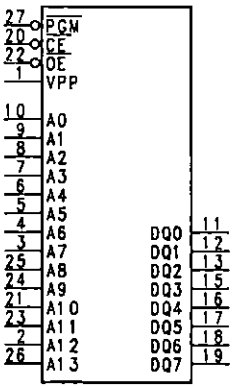
DRAWN BY: J P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY:	DATE 1-3-90		
APPROVED BY:	DATE 1-3-90	TITLE R* (PROMS/RAMS)	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 28 OF 113



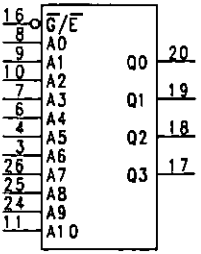
RE128K8-A



RE16K8

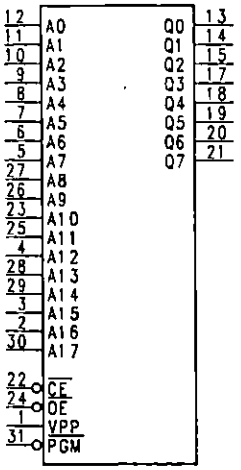


RE16K8-A

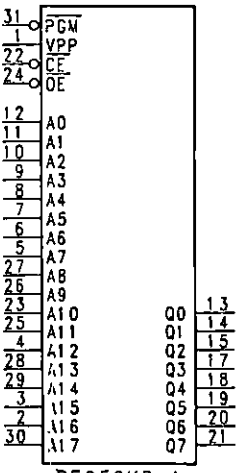


RE2048K4

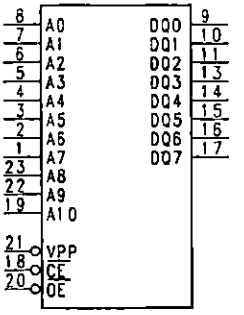
DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE R* (PROMS/RAMS)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 29 OF 113



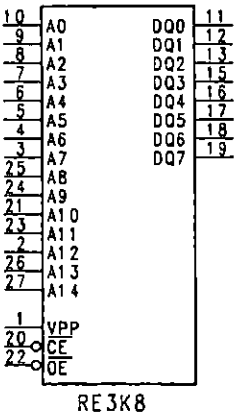
RE256K8



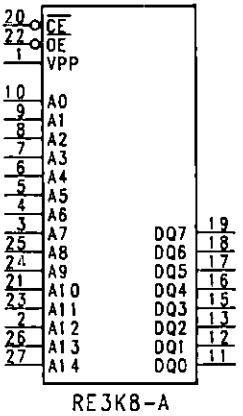
RE256K8-A



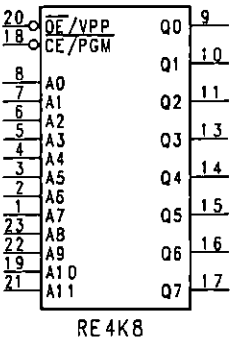
RE2K8



RE3K8

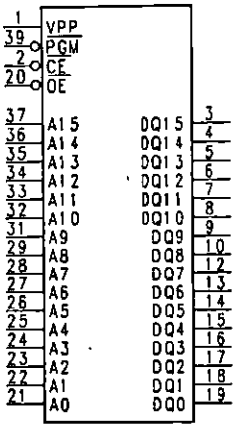


RE3K8-A

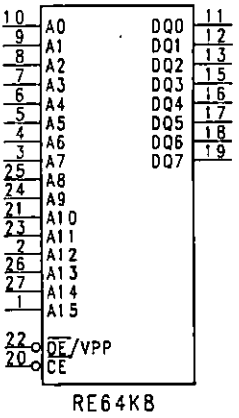


RE4K8

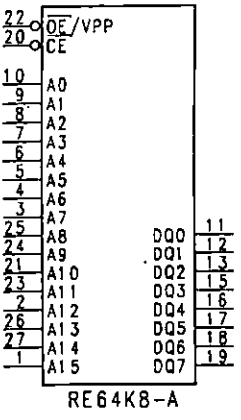
DRAWN BY: F.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: TP	DATE 1-3-90		
APPROVED BY:	DATE 1-3-90	TITLE R* (PROMS/RAMS)	
REVISION:	DATE		
JOB NAME:		REV. A	SCALE 1:1	SHEET 30 OF 113



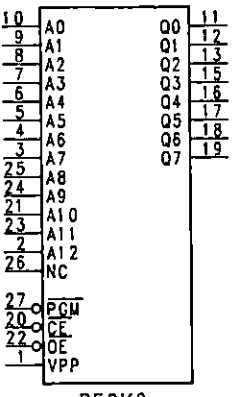
RE64K16



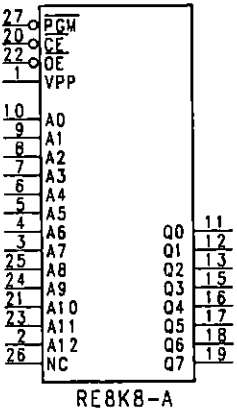
RE64K8



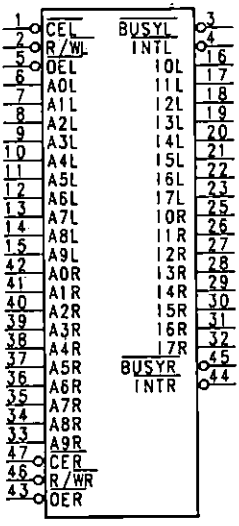
RE64K8-A



RE8K8

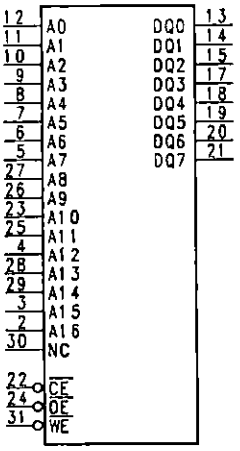


RE8K8-A

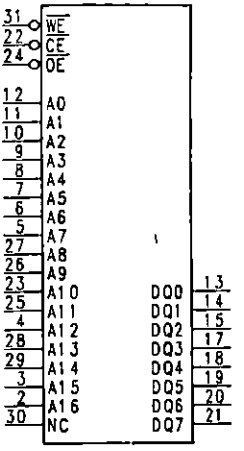


RS1024K8

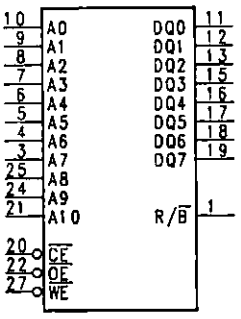
DRAWN BY: f P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY:	DATE 1-3-90		
APPROVED BY:	DATE 1-3-90	TITLE R* (PROMS/RAMS)	
REVISION:	DATE		
JOB NAME:		REV. A	SCALE 1:1	SHEET 31 OF 113



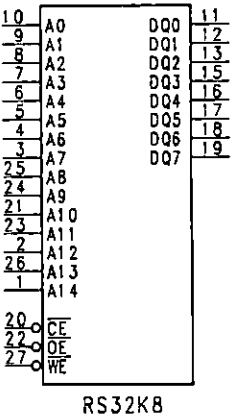
RS128K8



RS128K8-A

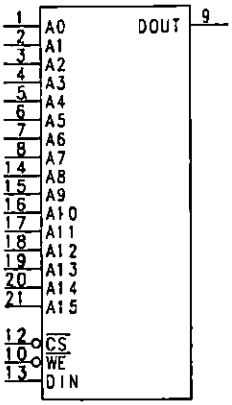


RS2K8

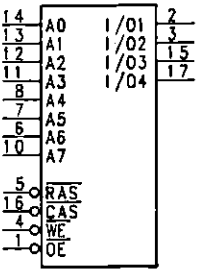


RS32K8

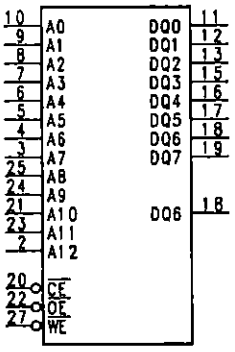
DRAWN BY: J P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: (illegible)	DATE 1-3-90			
APPROVED BY:	DATE	TITLE R* (PROMS/RAMS)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 32 OF 113



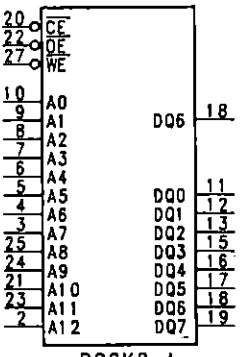
RS64K1



RS64K4

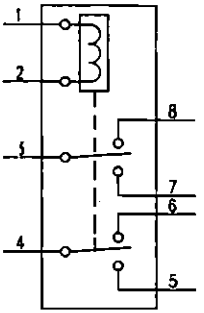


RS8K8

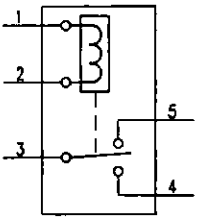


RS8K8-A

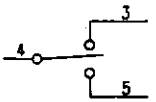
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY:	DATE	TITLE R* (PROMS/RAMS)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 33 OF 113



RLY-DPDT



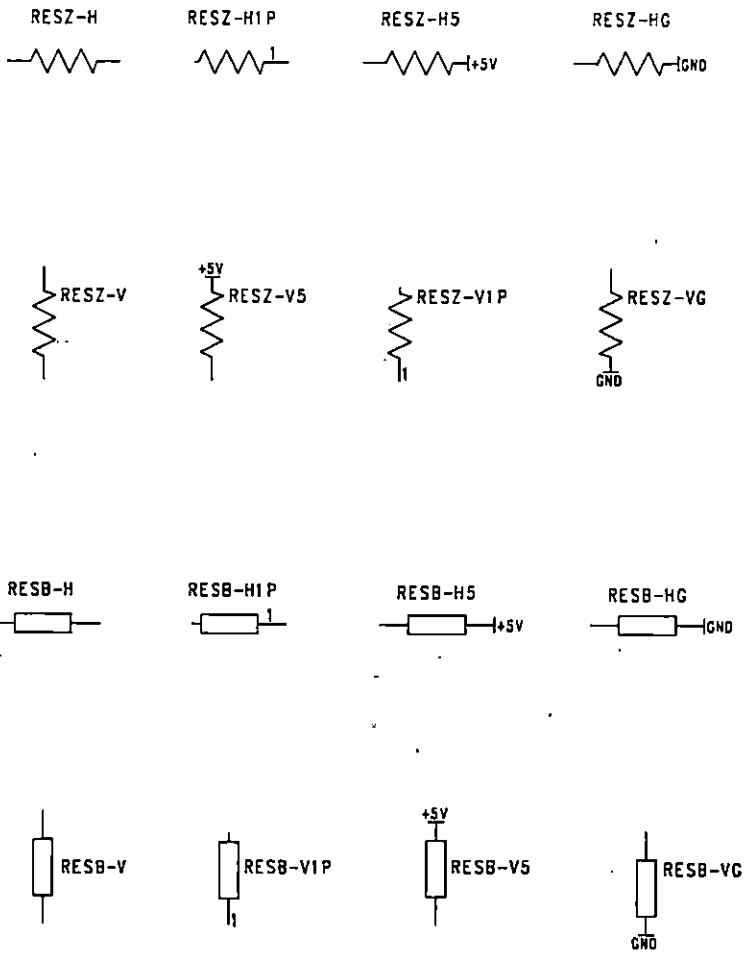
RLY-SPDT



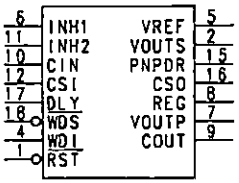
RLY-SW-SPDT

RLY-COIL

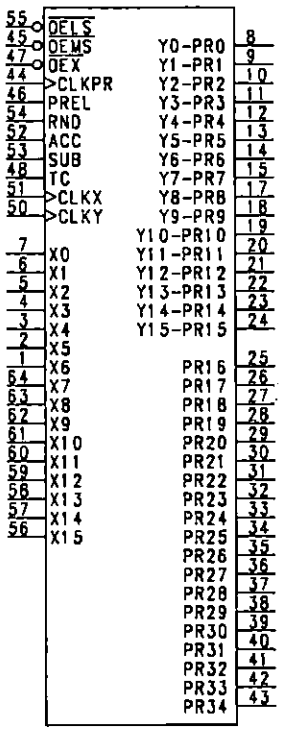
DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3 90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE RELAYS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 34 OF 113



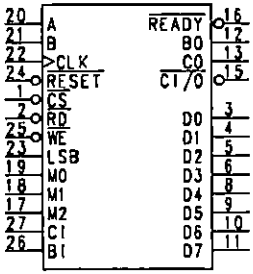
DRAWN BY: L.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DP	DATE 1-3-90			
APPROVED BY:	DATE	TITLE RESISTORS		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 35 OF 113



TCA5600

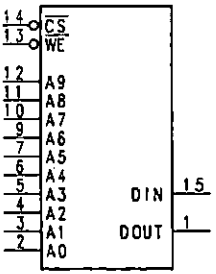


THCT1010

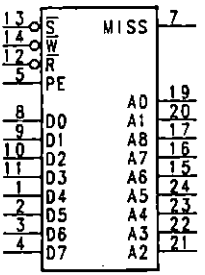


THCT2000E

DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DO	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE T******		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 36 OF 113

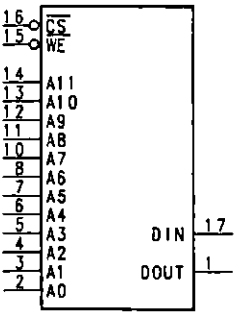


100415

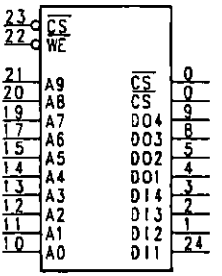


100469

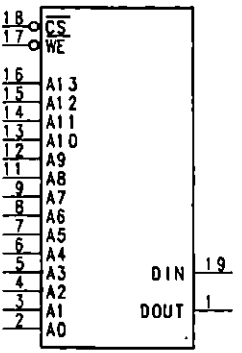
DRAWN BY: J P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: TC	DATE 1-390			
APPROVED BY:	DATE 1-3-90	TITLE 1XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 37 OF 113



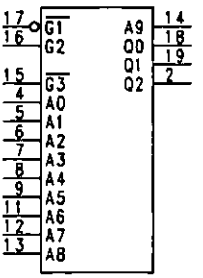
100470



100474

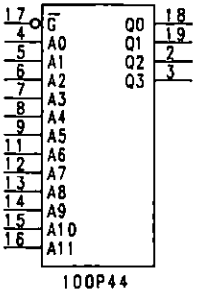


100480

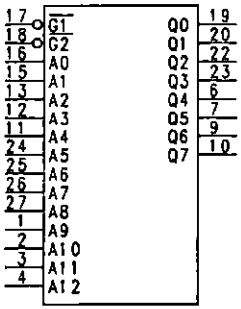


100P14

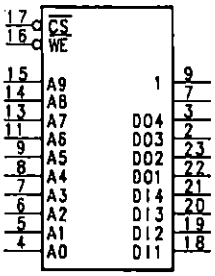
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DD	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 1XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 38 OF 113



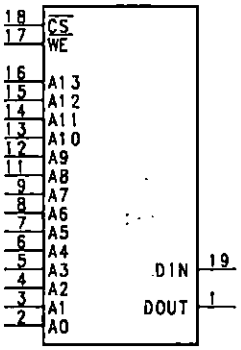
100P44



100P88

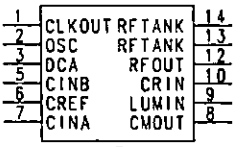


10474

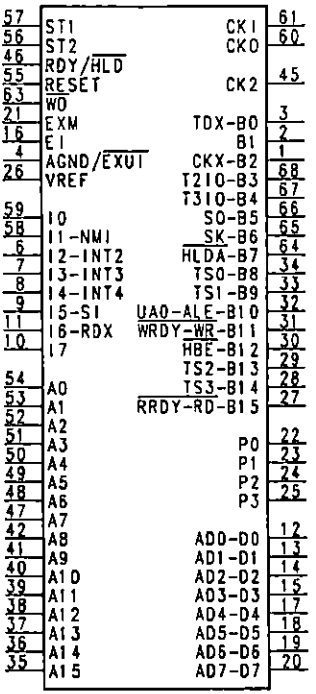


10480

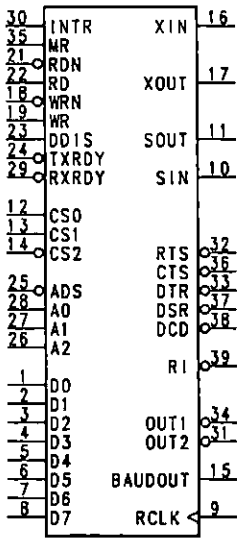
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 1XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 39 OF 113



1372

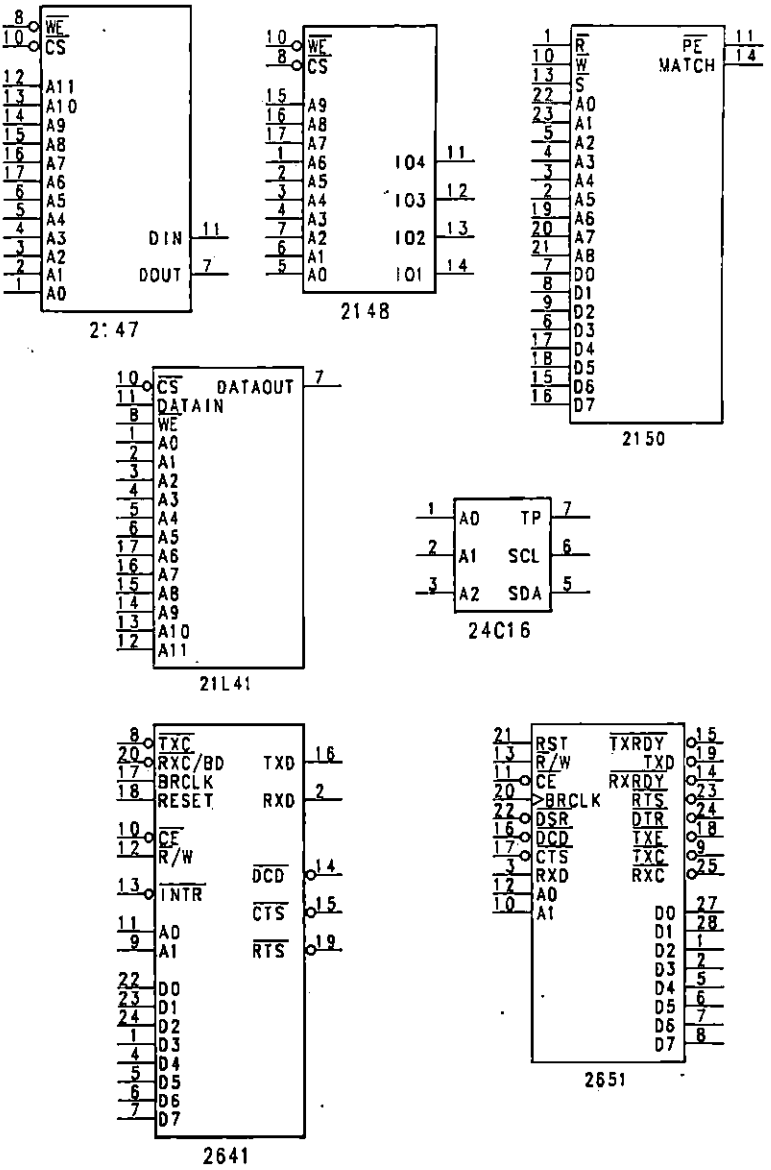


16164

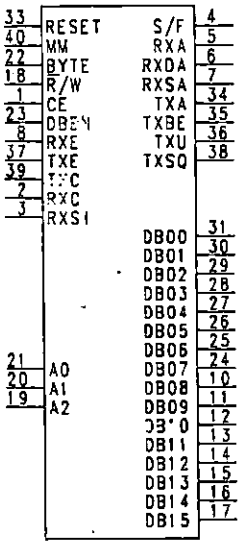


16550

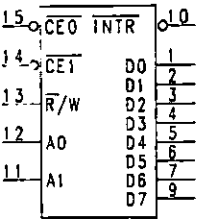
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: RD	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 1XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 40 OF 113



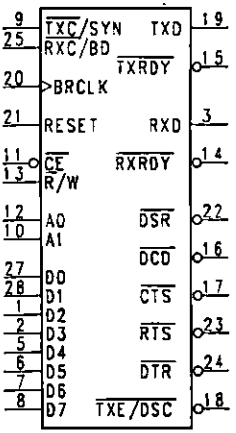
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 2XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 41 OF 113



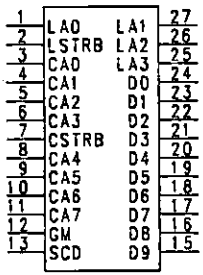
2652



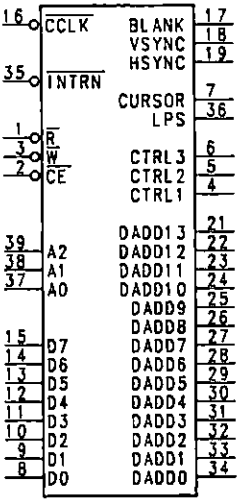
2653



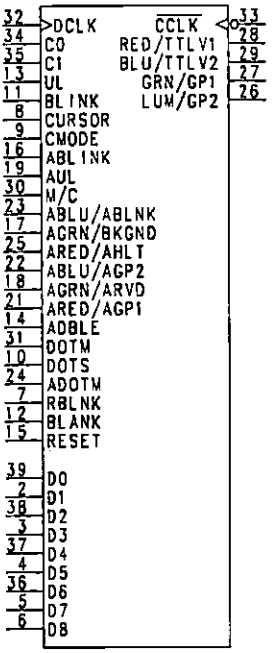
2661



2670

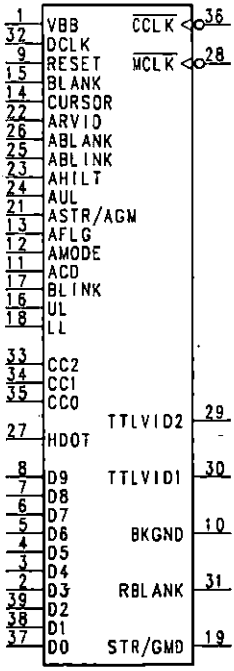


2672

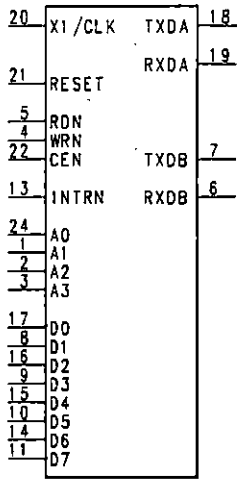


2675T

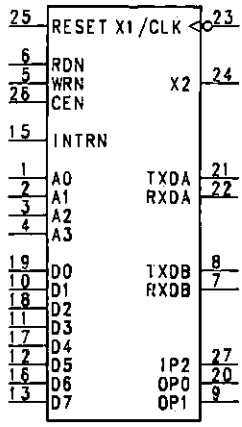
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: RC	DATE 1-3-90		
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 2XXXXXX	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 42 OF 113



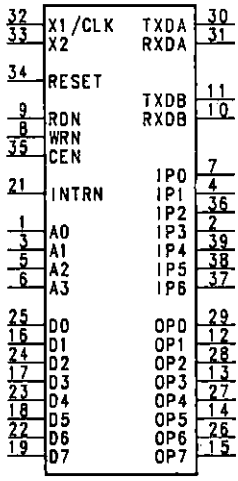
2677



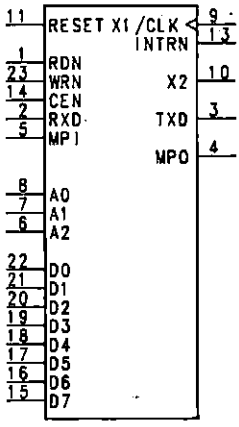
2681N24



2681N28

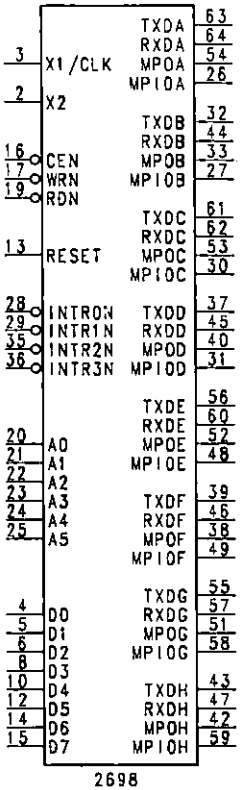


2681N40

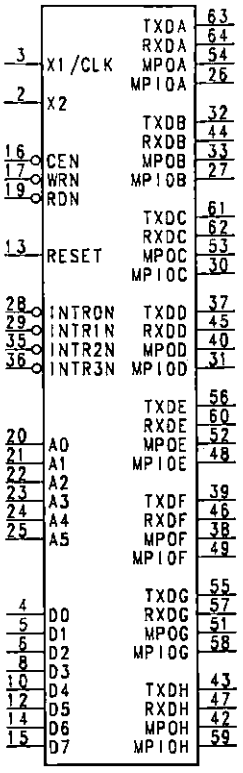


2691

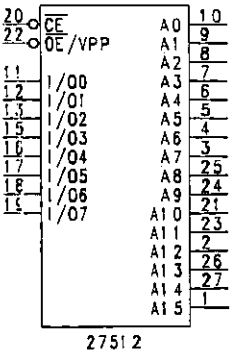
DRAWN BY: L.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: RD	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 2XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 43 OF 113



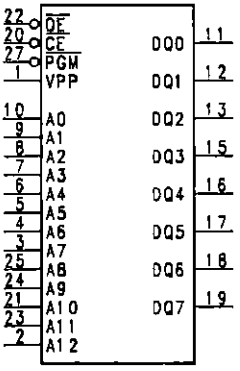
2698



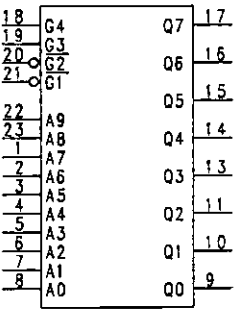
2698SM



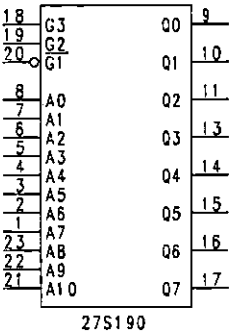
27512



2764

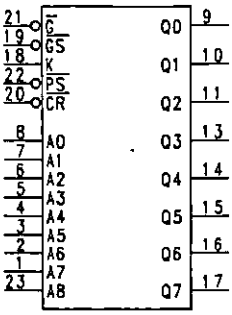


27S180

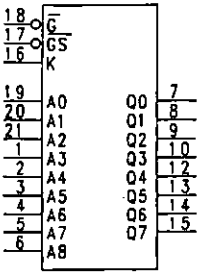


27S190

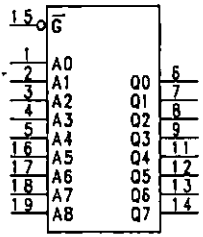
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: RC	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 2XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 44 OF 113



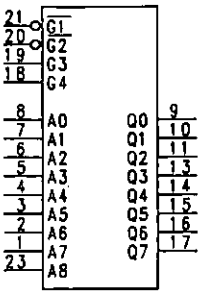
27S25



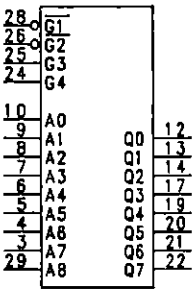
27S27



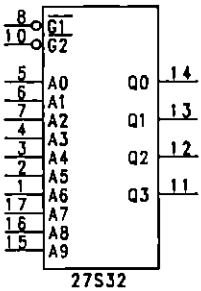
27S28



27S31

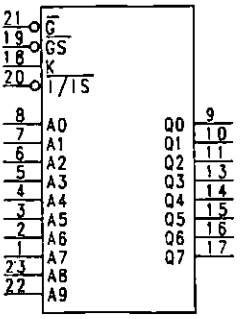


27S31L

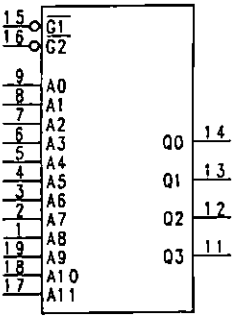


27S32

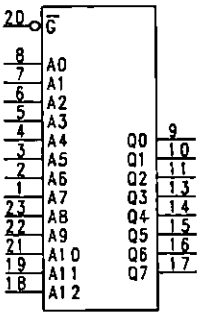
DRAWN BY: L.B.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 2XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 45 OF 113



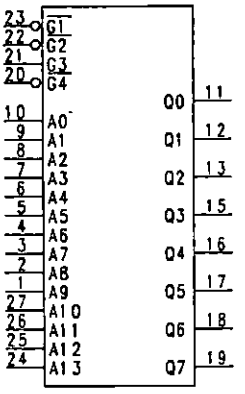
27S35



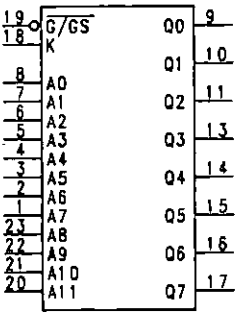
27S41



27S49

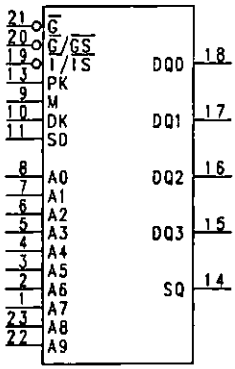


27S51

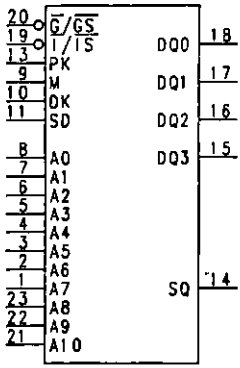


27S55

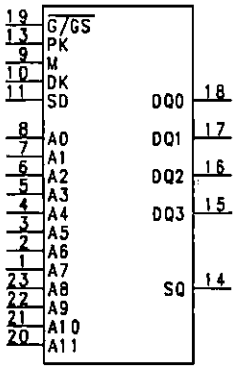
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 2XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 46 OF 113



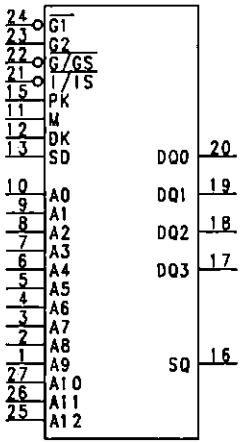
27S65



27S75

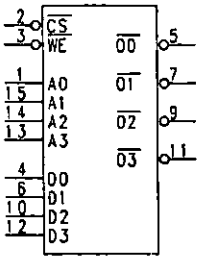


27S85

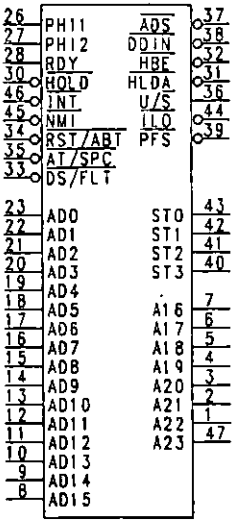


27S95

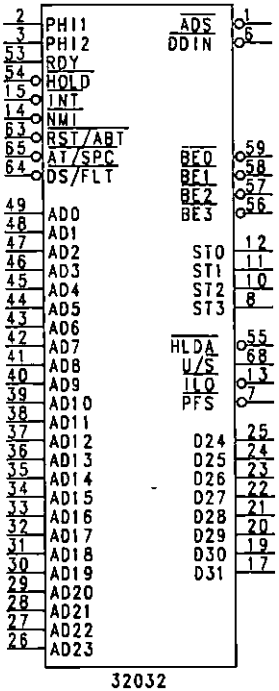
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 2XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 47 OF 113



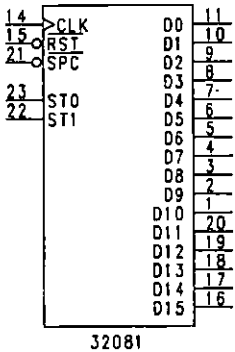
31L01



32016

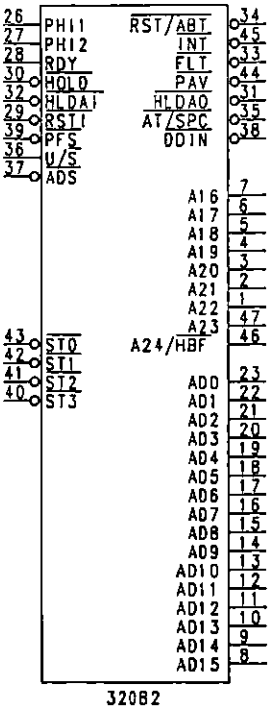


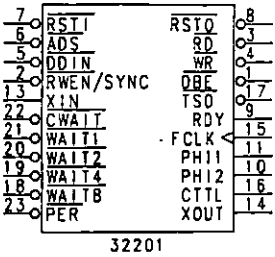
32032

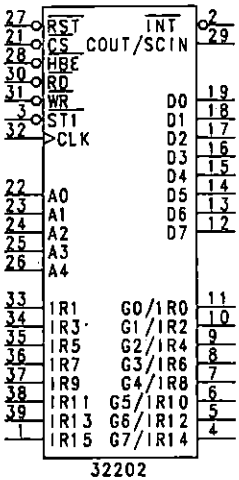


32081

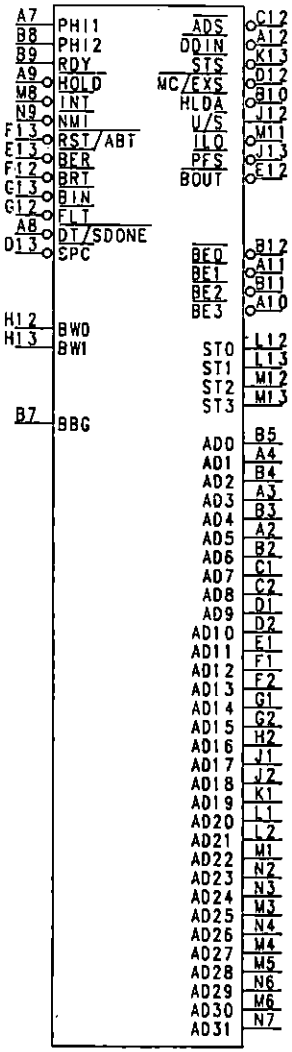
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 3XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 48 OF 113



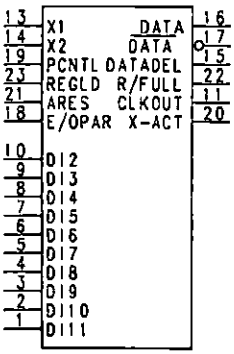




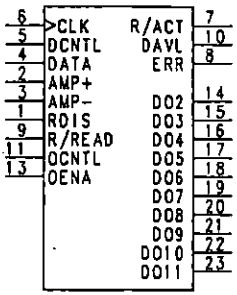
DRAWN BY: L.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 3XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 49 OF 113



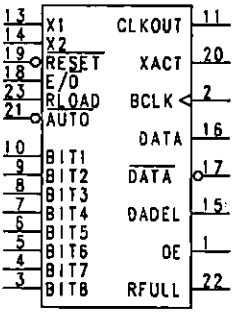
32332



32440

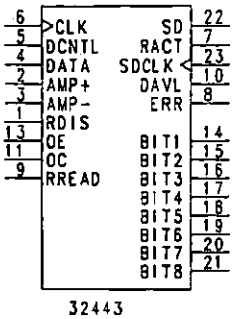


32441

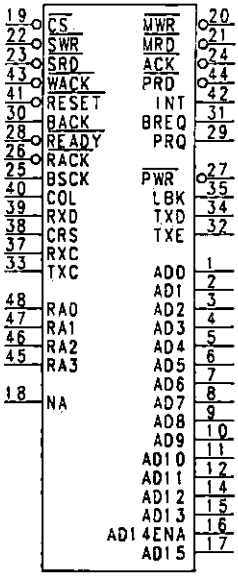


32442

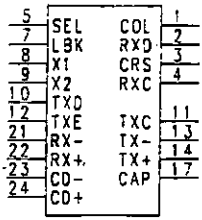
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 3XXXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 50 OF 113



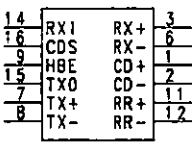
32443



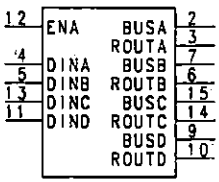
32490



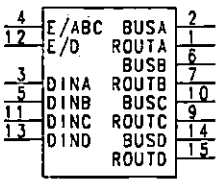
32491



32492

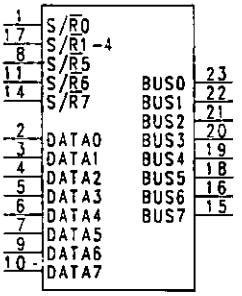


3440

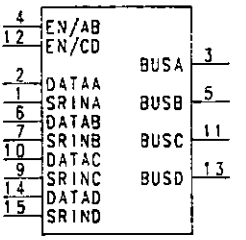


3446

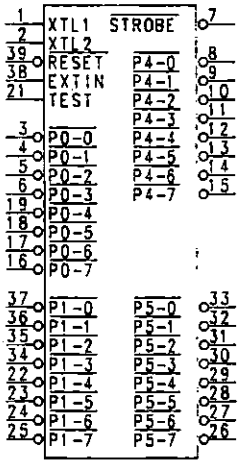
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: (RP)	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 3XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 51 OF 113



3447

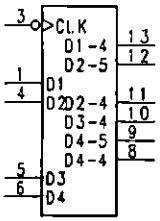


3448

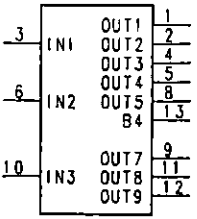


3870

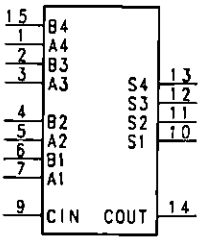
DRAWN BY: L P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 3XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 52 OF 113



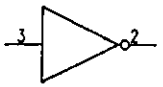
4006

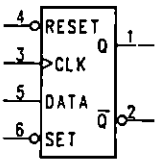


4007

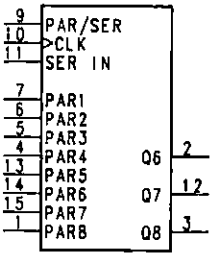


4008

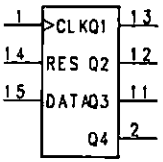

4009



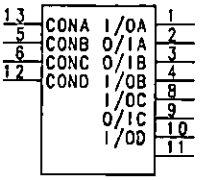
4013



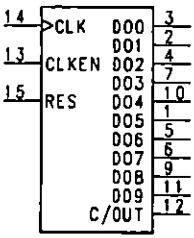
4014



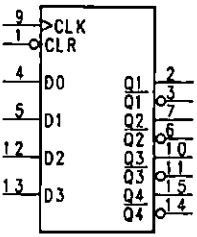
4015



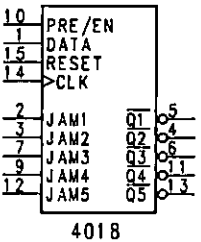
4016



4017

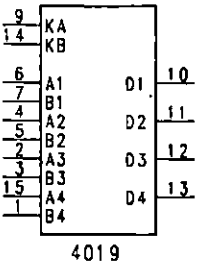


40175

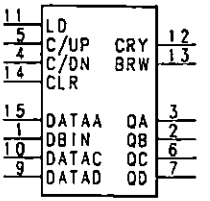


4018

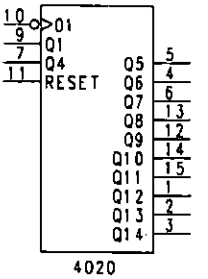
DRAWN BY: J.F.	DATE 1.3.90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1.3.90			
APPROVED BY:	DATE 1-3-90	TITLE 4XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 53 OF 113



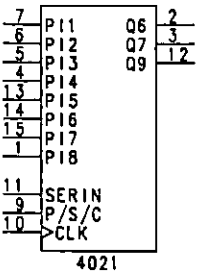
4019



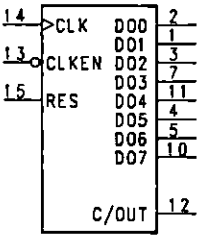
40193



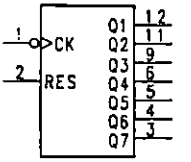
4020



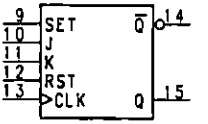
4021



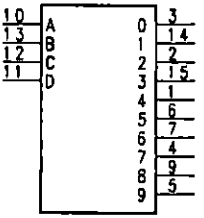
4022



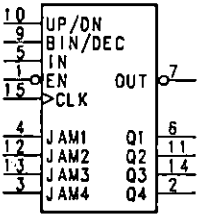
4024



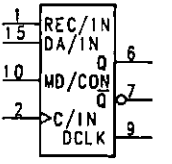
4027



4028

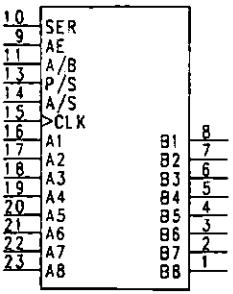


4029

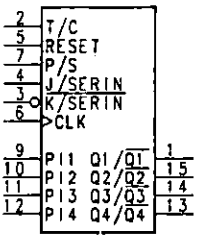


4031

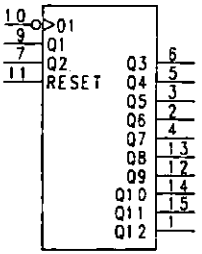
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: RC	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 4XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 54 OF 113



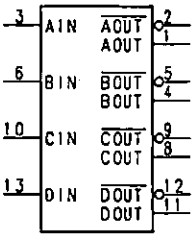
4034



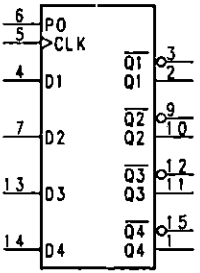
4035



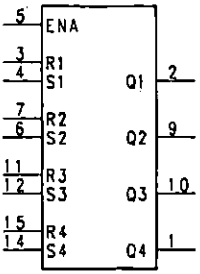
4040



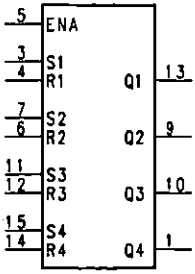
4041



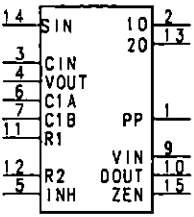
4042



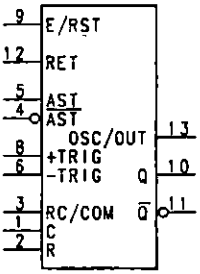
4043



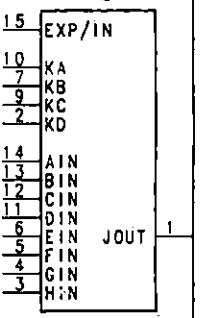
4044



4046

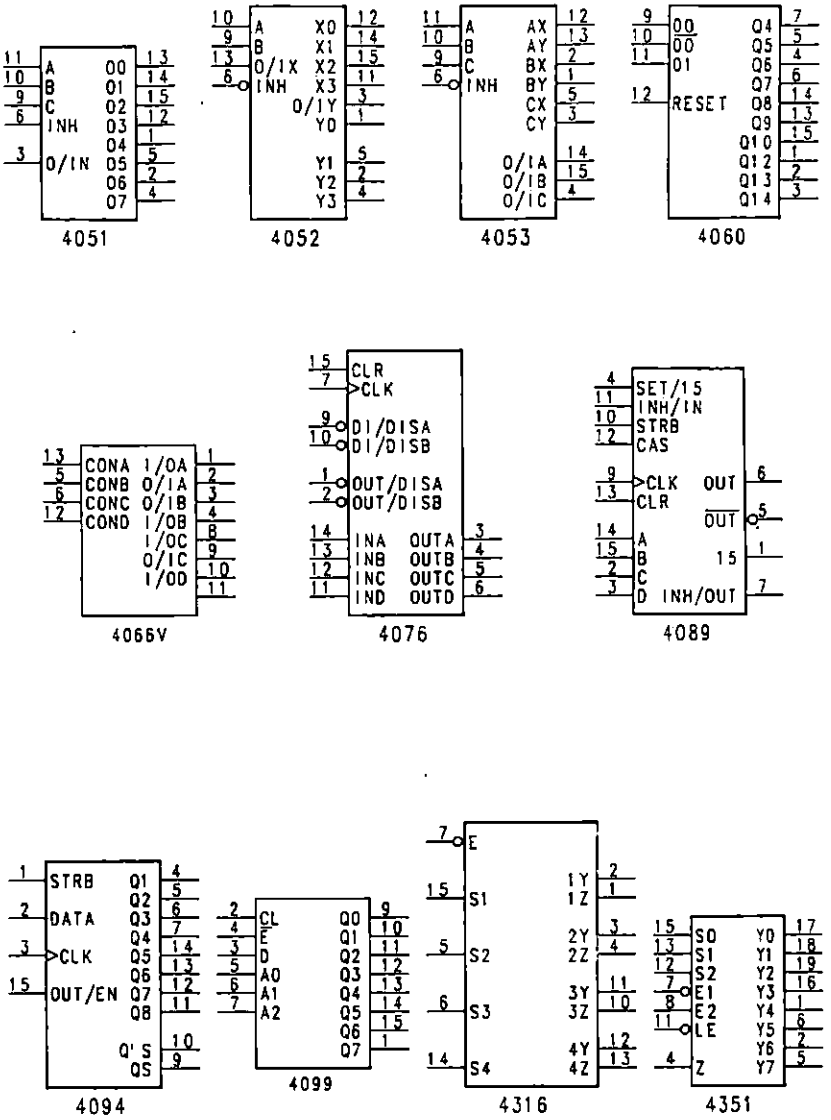


4047

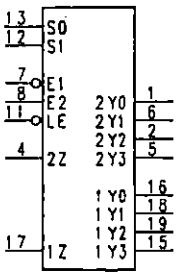


4048

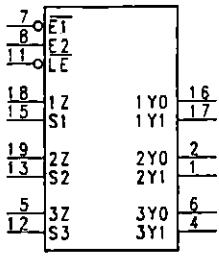
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-9-90	TITLE 4XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 55 OF 113



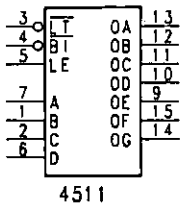
DRAWN BY: P. P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY:	DATE 1.3.90		
APPROVED BY:	DATE 1-9-90	TITLE 4XXXXXX	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 56 OF 113



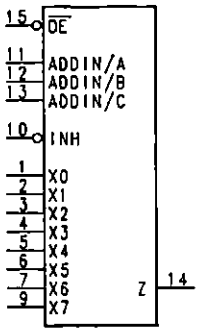
4352



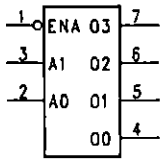
4353



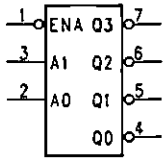
4511



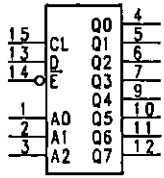
4512



4555

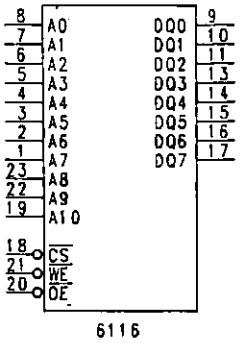


4556

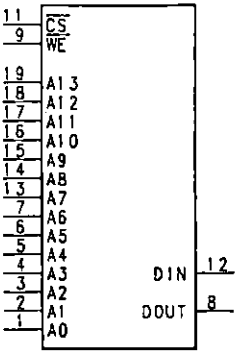


4724

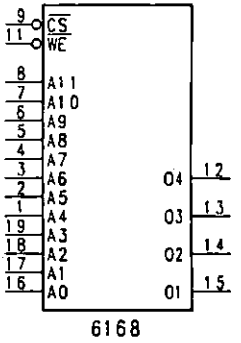
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1.3.90			
APPROVED BY:	DATE 1-3-90	TITLE 4XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 57 OF 113



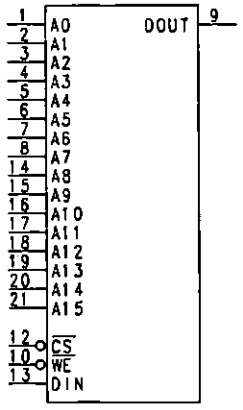
6116



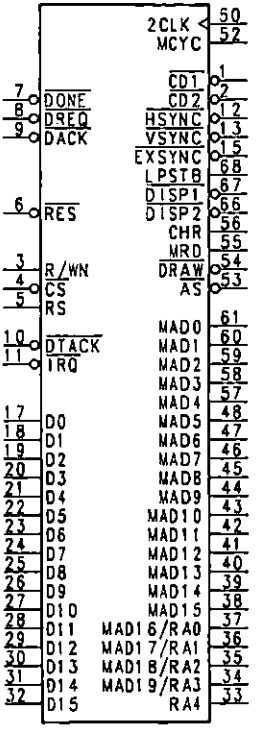
6167



6168

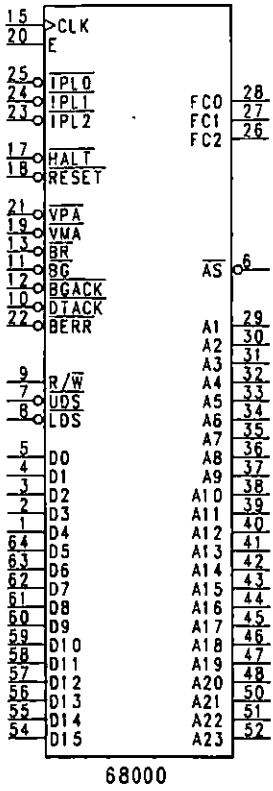


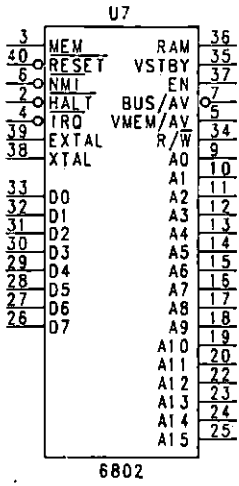
6287

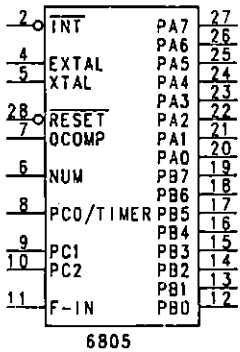


63484

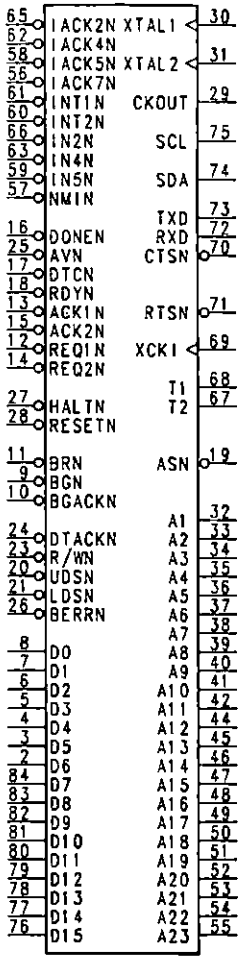
DRAWN BY: [illegible]	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: [illegible]	DATE 1.390		
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 6XXXXXX	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 58 OF 113

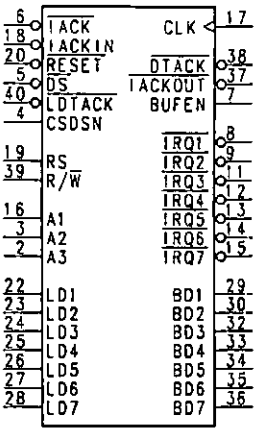


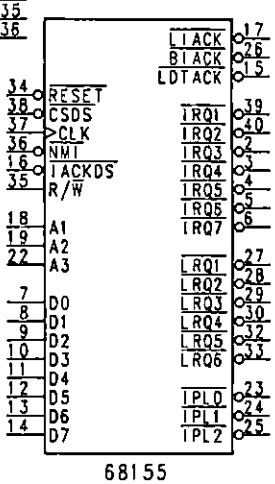


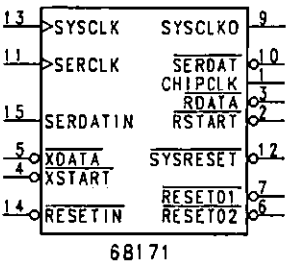


DRAWN BY: F.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1.3.90			
APPROVED BY:	DATE 1-3-90	TITLE 6XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 59 OF 113

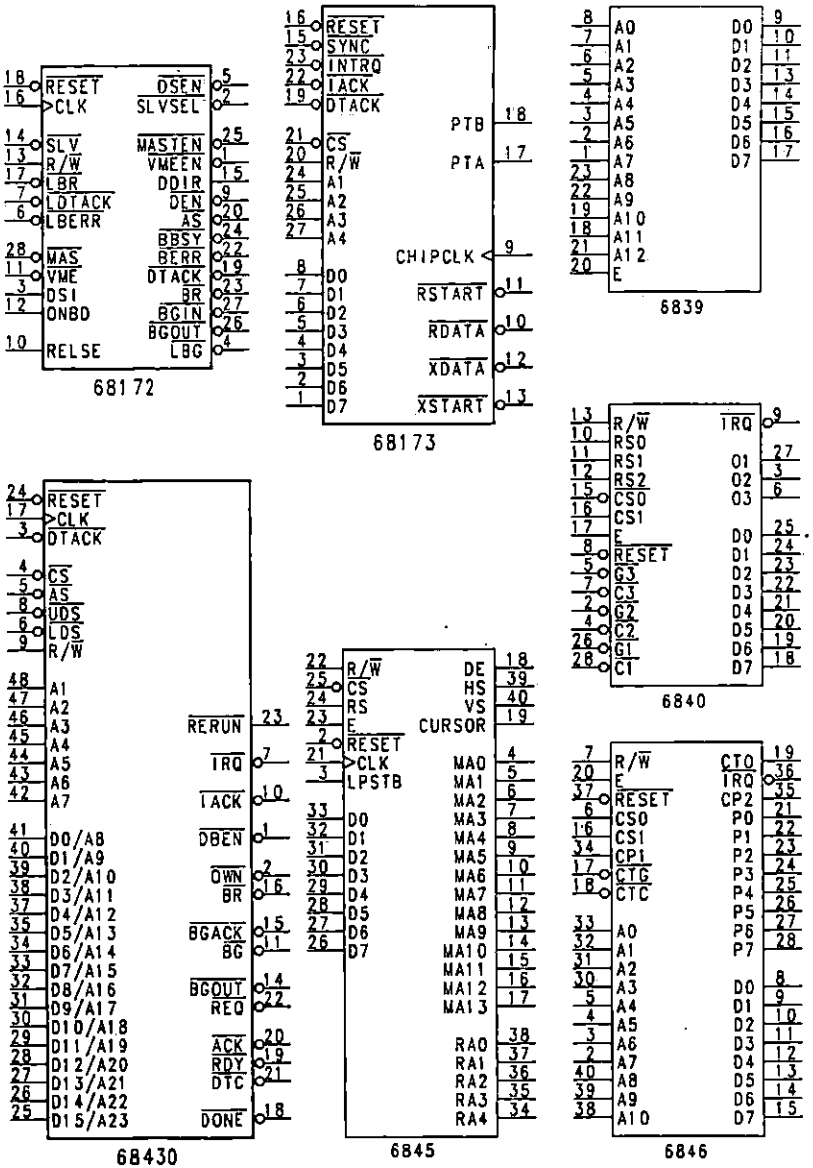




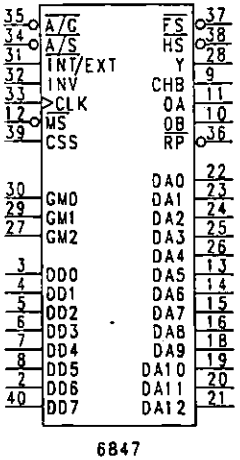




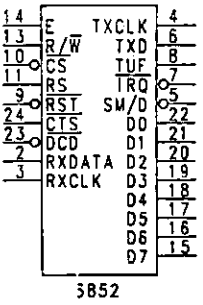
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 6XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 60 OF 113



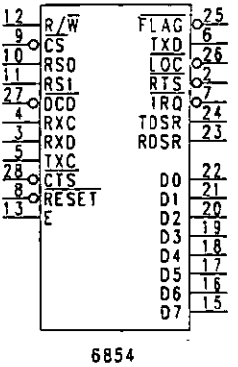
DRAWN BY: L.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY:	DATE 1-3-90		
APPROVED BY:	DATE 1-3-90	TITLE 6XXXXXX	
REVISION:	DATE		
JOB NAME:		REV. A	SCALE 1:1	SHEET 61 OF 113



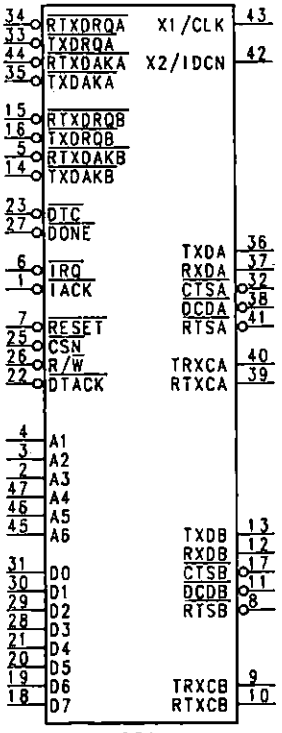
6847



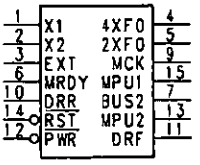
3852



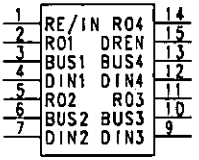
6854



68562

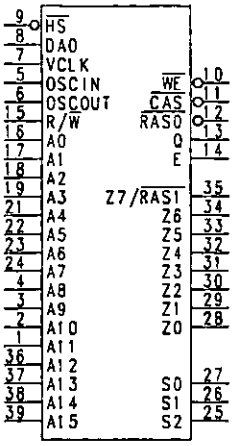


6875

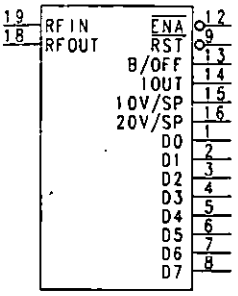


6880A

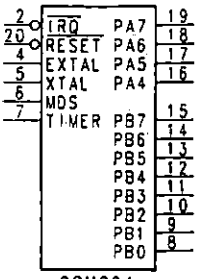
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 6XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 62 OF 113



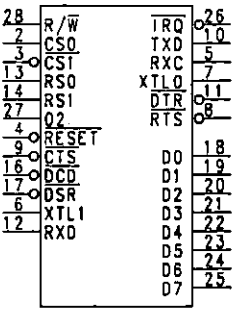
6883



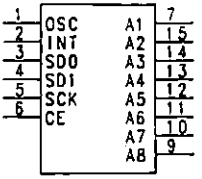
6890



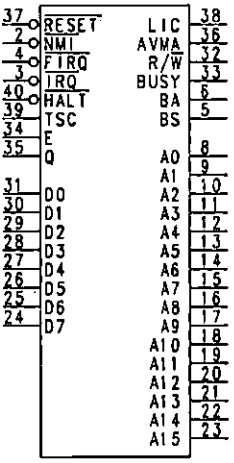
68HC04



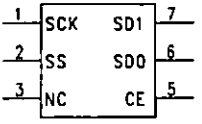
68HC51



68HC68

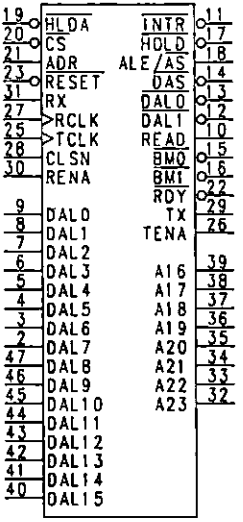


68HC09

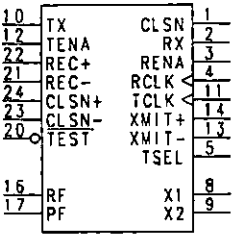


68HC68R1

DRAWN BY: J P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1.3.90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 6XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 63 OF 113

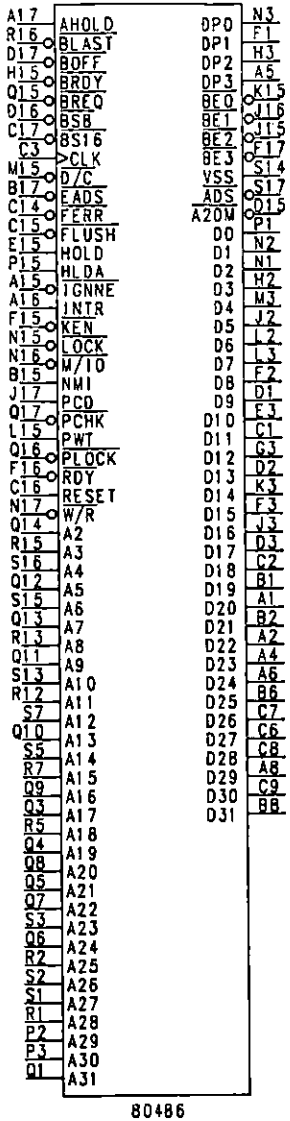


7990

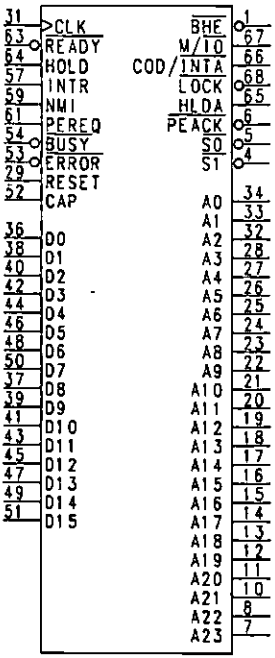


7992

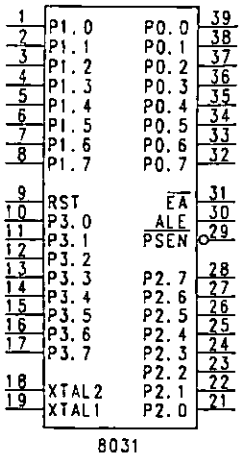
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 7XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 64 OF 113



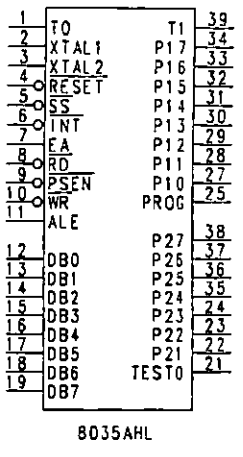
DRAWN BY: J P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 65 OF 113



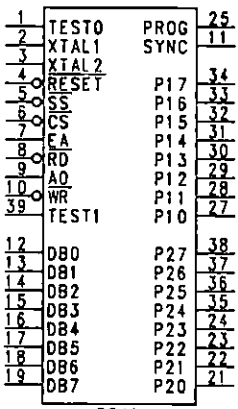
80286



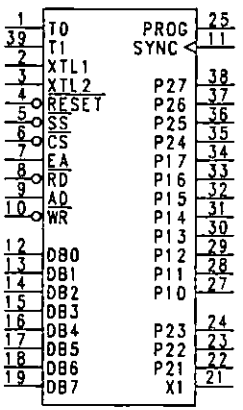
8031



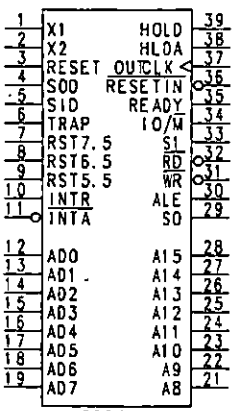
8035AHL



8041

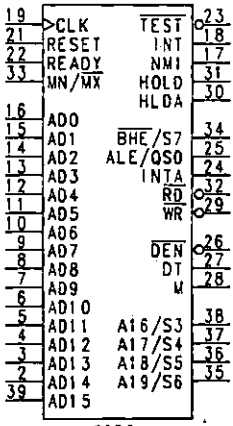


8042

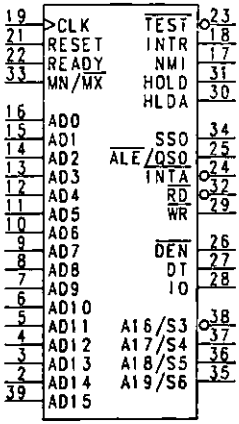


8085AH

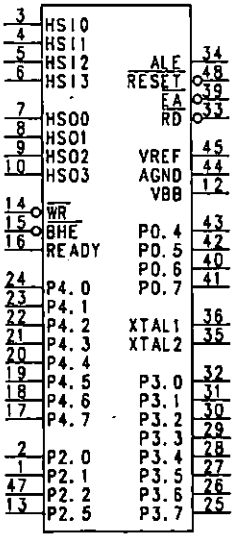
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: RD	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 66 OF 113



8086

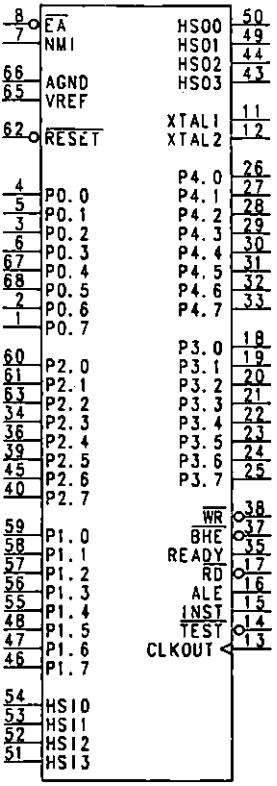


8088

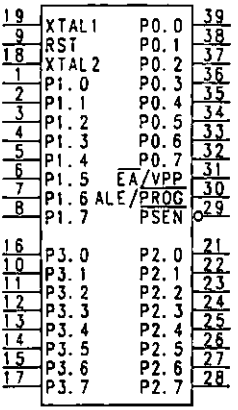


8094

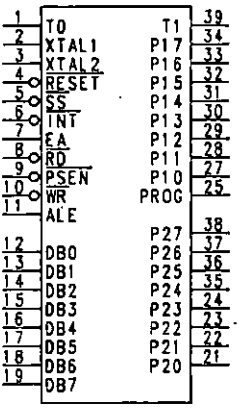
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: RD	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 67 OF 113



8096

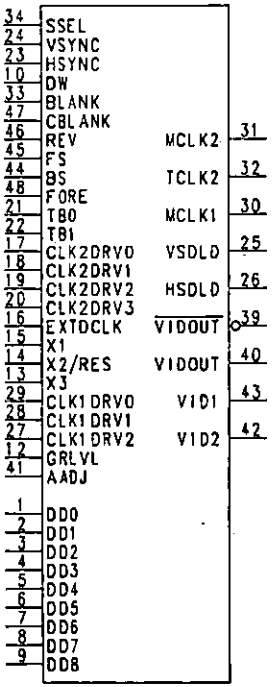


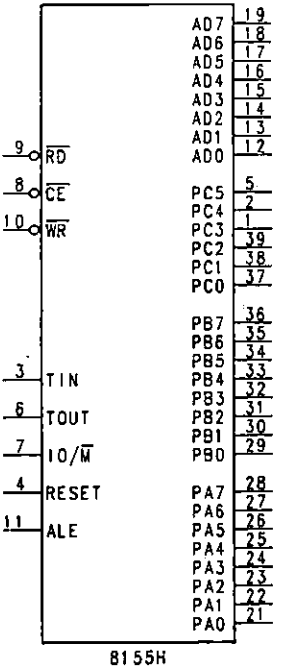
80C252

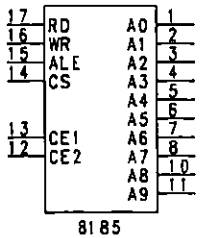


80C39

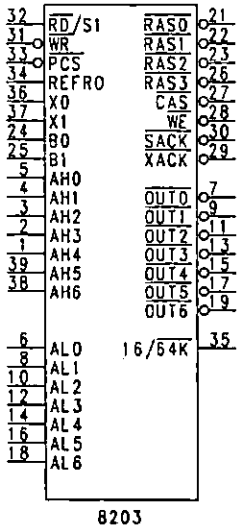
DRAWN BY: L.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 68 OF 113



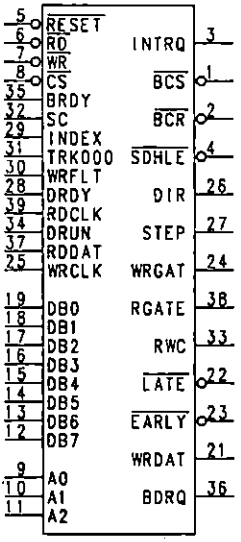




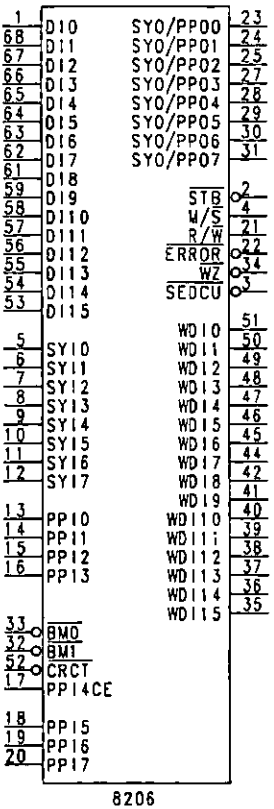
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 69 OF 113



8203

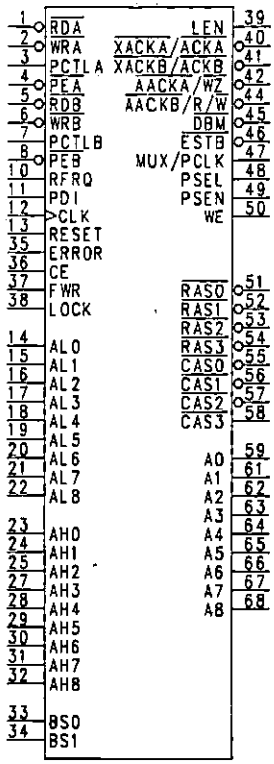


B2064

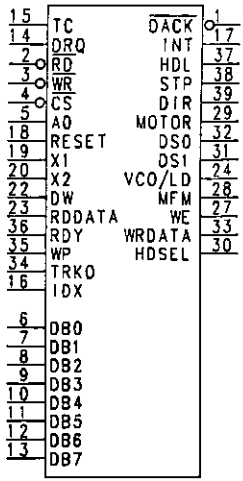


8206

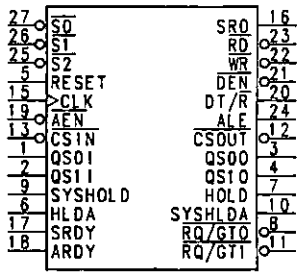
DRAWN BY: [illegible]	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 70 OF 113



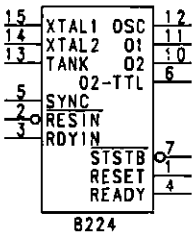
8207



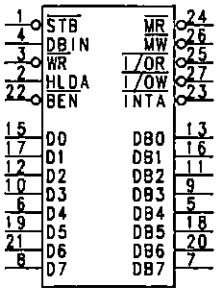
82072



82188

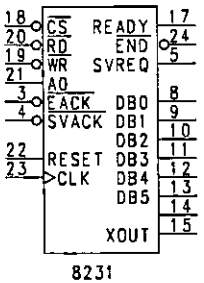


8224

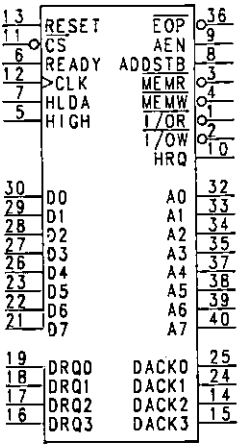


8228

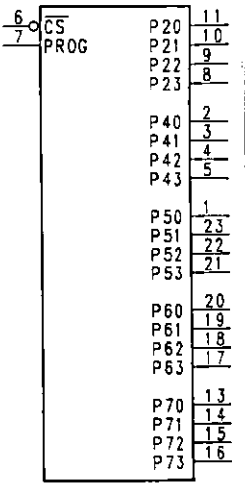
DRAWN BY: L.P.	DATE 1.3.90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: DC	DATE 1.3.90		
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXX	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 71 OF 113



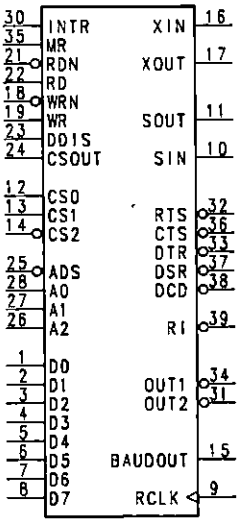
8231



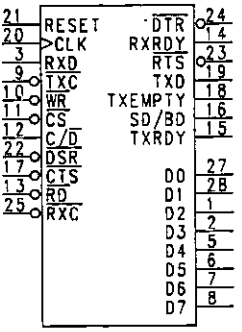
8237A



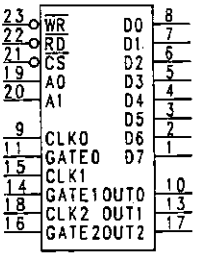
8243



8250

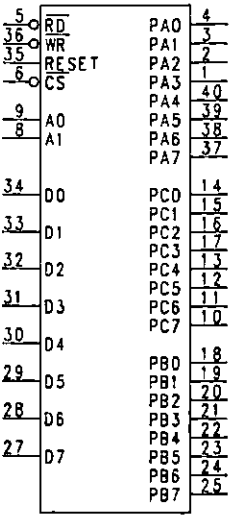


8251A

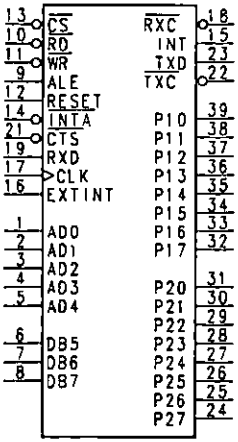


8253

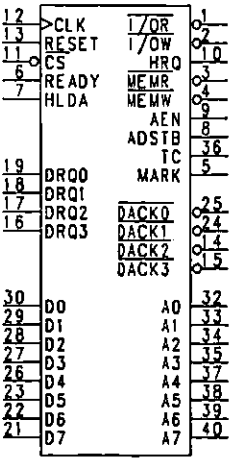
DRAWN BY: J.P.	DATE 1.2.90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: (illegible)	DATE 1-3-90		
APPROVED BY: (illegible)	DATE 1-3-90	TITLE 8XXXXXX	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 72 OF 113



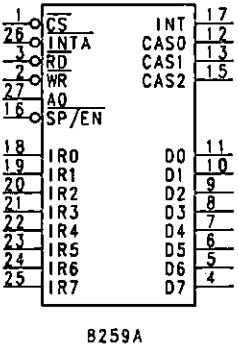
8255A



8256AH

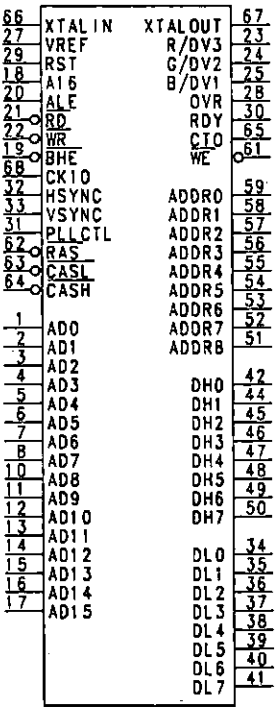


8257

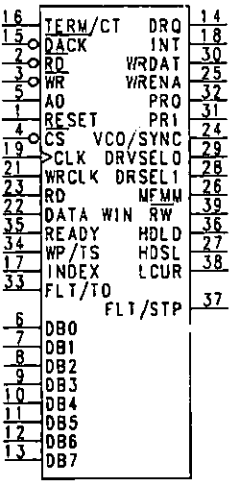


8259A

DRAWN BY: L P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1.3.90			
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 73 OF 113

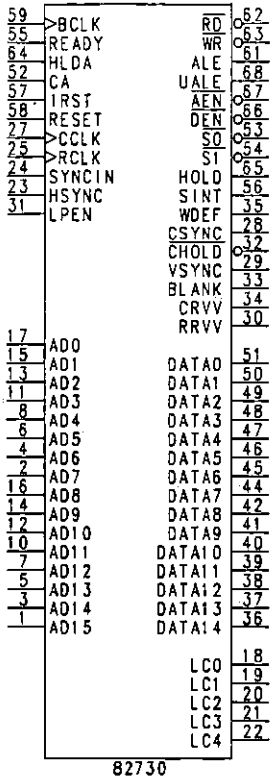


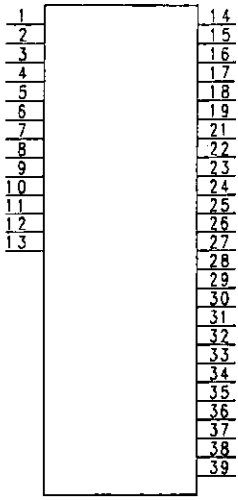
82716

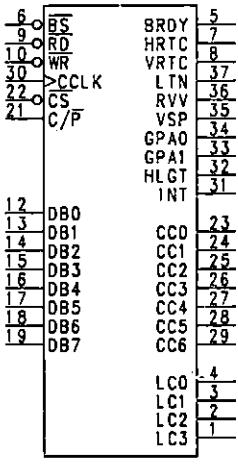


8272A

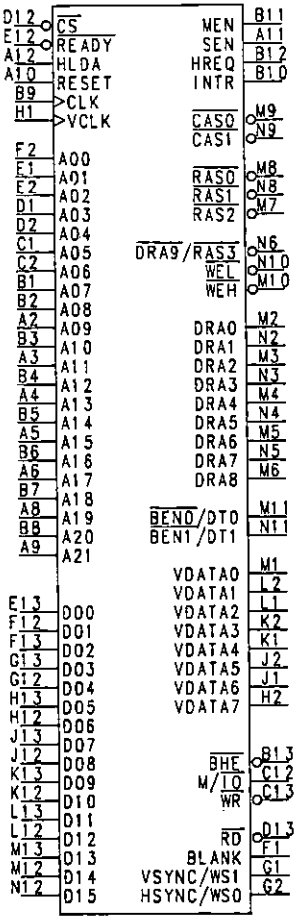
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DD	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 74 OF 113



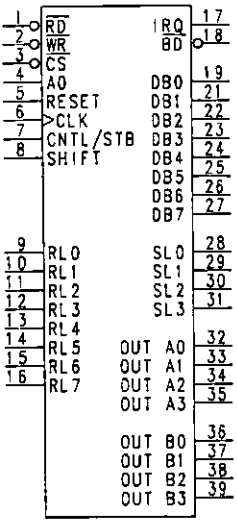




DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1.3.90			
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 75 OF 113

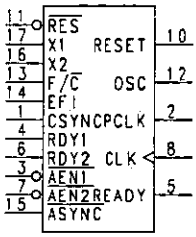


82786

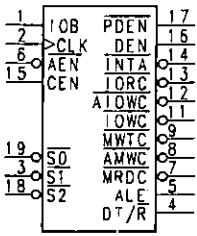


8279

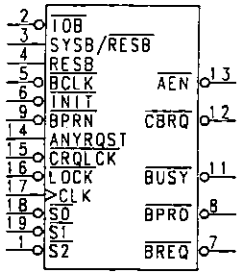
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: RD	DATE 1.3.90			
APPROVED BY: LH	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 76 OF 113



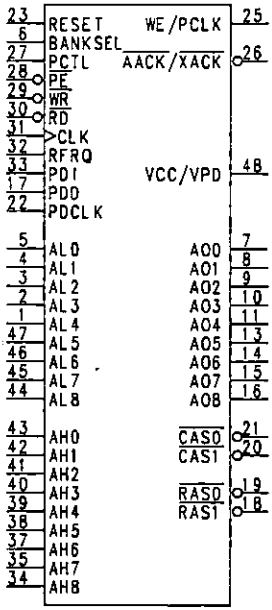
8284



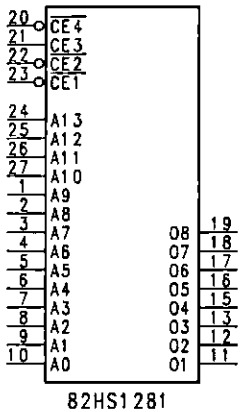
8288



8289

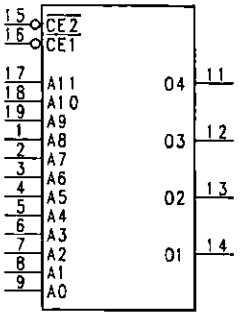


82C08

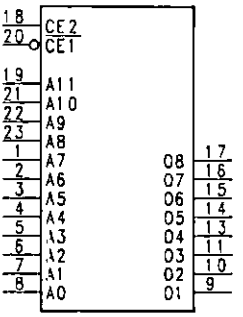


82HS1281

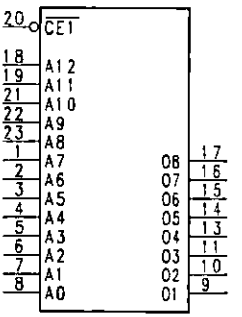
DRAWN BY: L.D.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 77 OF 113



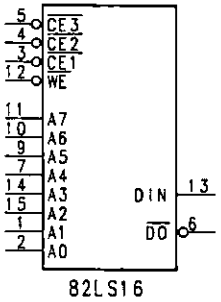
82HS195



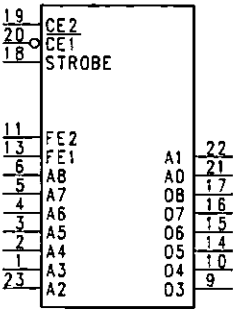
82HS321



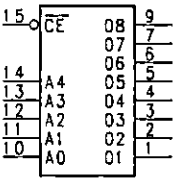
82HS641



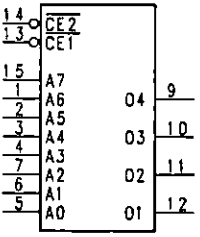
82LS16



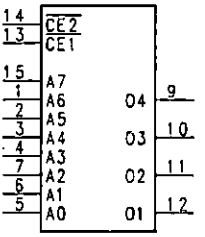
82S115



82S123

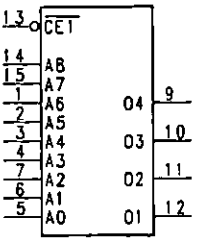


82S126

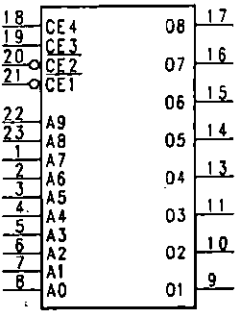


82S129

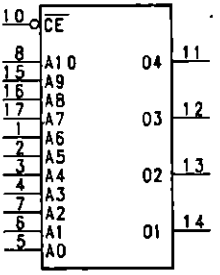
DRAWN BY: L.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 8XXXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 78 OF 113



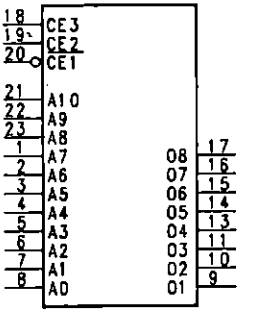
82S130



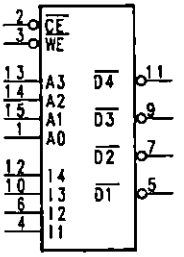
82S181



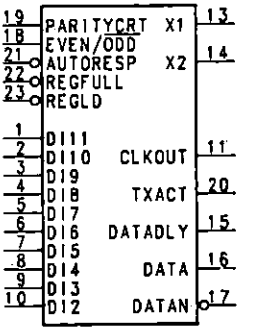
82S185



82S191

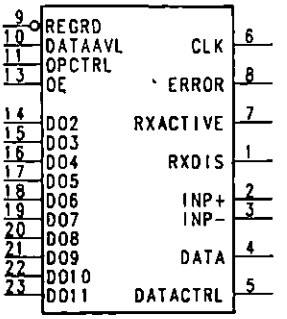


82S25

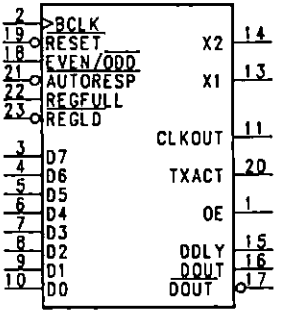


8340

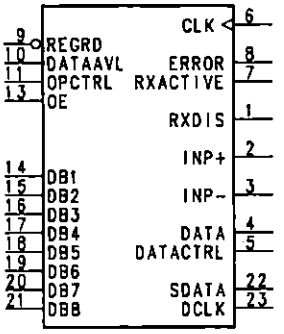
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 79 OF 113



8341

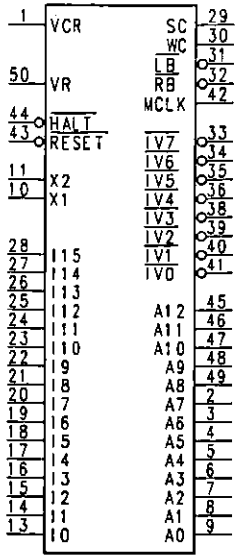


8342

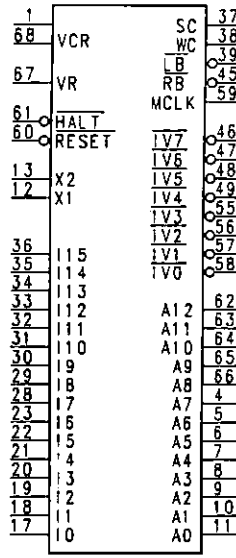


8343

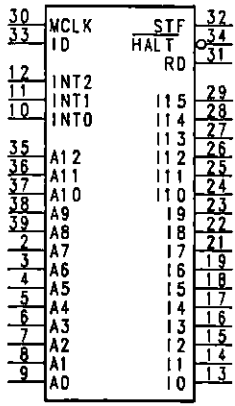
DRAWN BY: KP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DP	DATE 1.3.90			
APPROVED BY: Mark	DATE 1-3-90	TITLE 8XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 80 OF 113



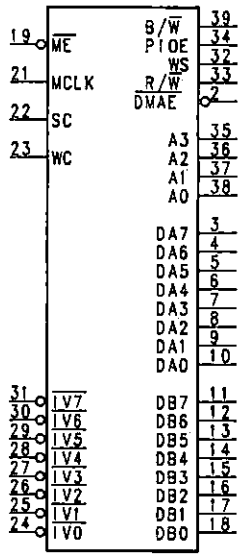
8X305



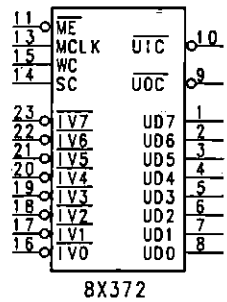
8X305-PLCC



8X310

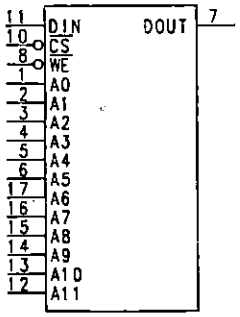


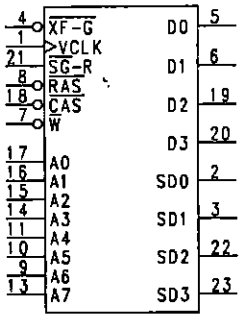
8X320

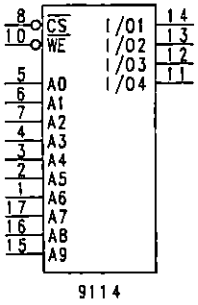


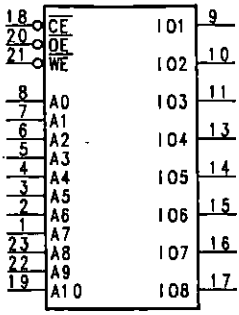
8X372

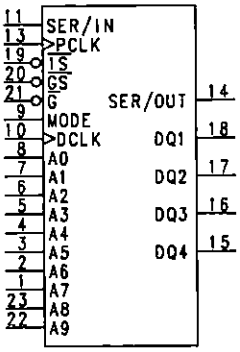
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: (illegible)	DATE 1.3.90		
APPROVED BY: (illegible)	DATE 1-3-90	TITLE 8XXXXXX	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 81 OF 113

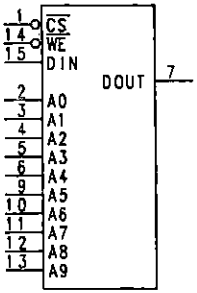


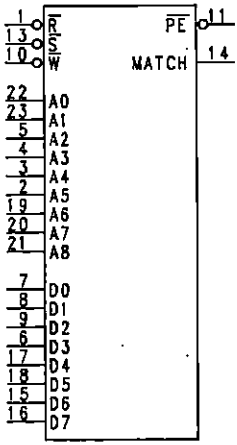


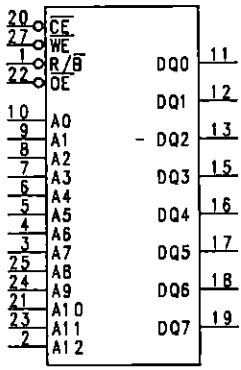


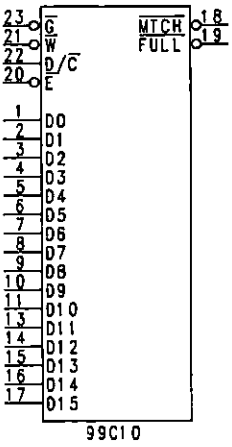




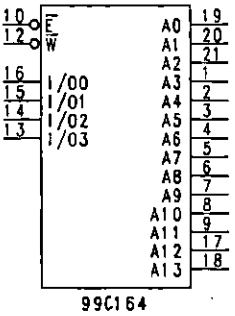




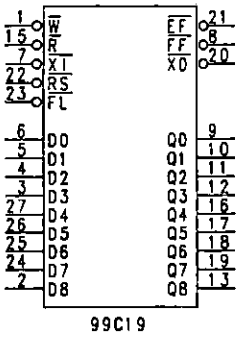




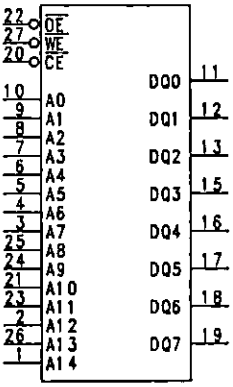
DRAWN BY: J P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1.3.90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 9XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 82 OF 113



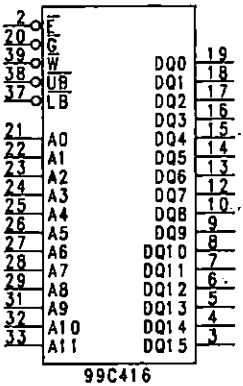
99C164



99C19

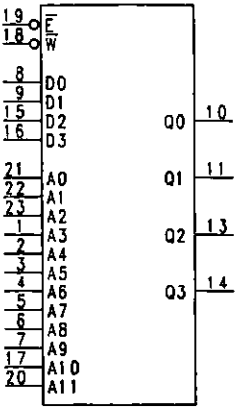


99C32B

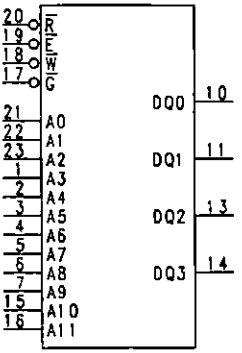


99C416

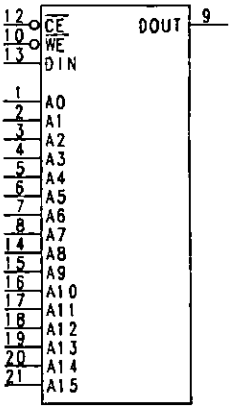
DRAWN BY: J P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1.3.90			
APPROVED BY:	DATE 1-3-90	TITLE 9XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 83 OF 113



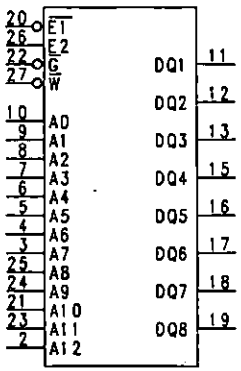
99C58



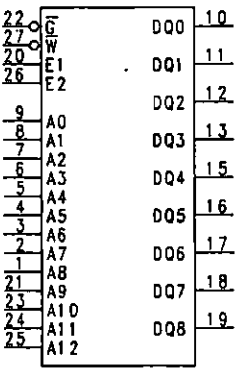
99C60



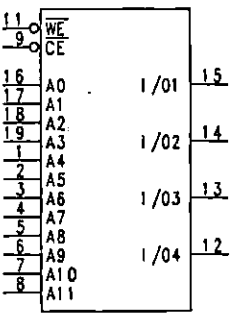
99C641



99C88

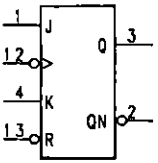


99C89

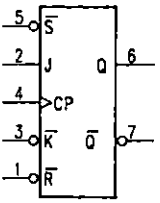


99CL68

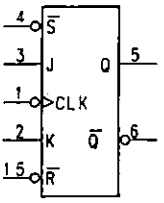
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: RO	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 9XXXXXX		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 84 OF 113



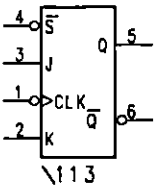
\107



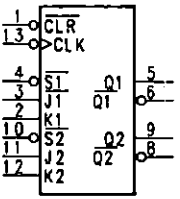
\109



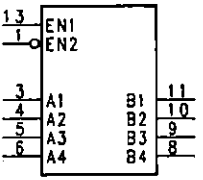
\112



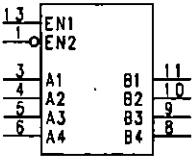
\113



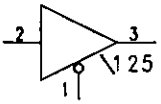
\114



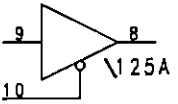
\1242



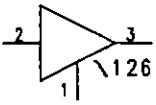
\1243



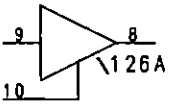
\125



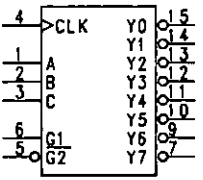
\125A



\126

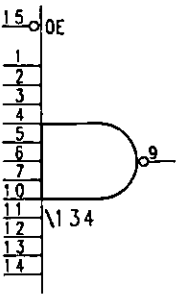


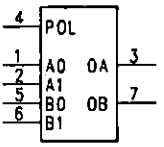
\126A

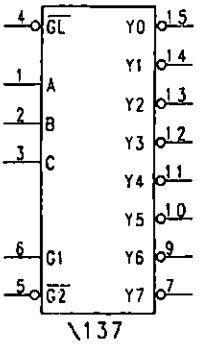


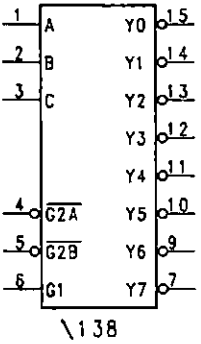
\131

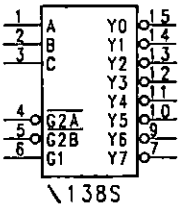
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: QC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 85 OF 113

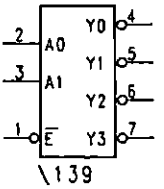


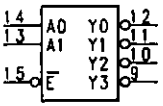


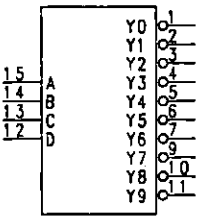




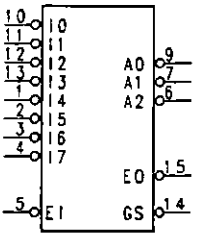




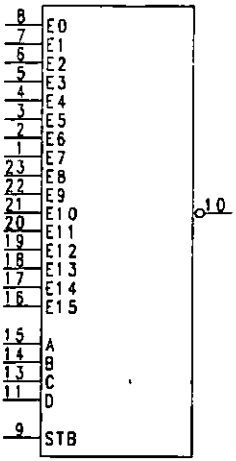




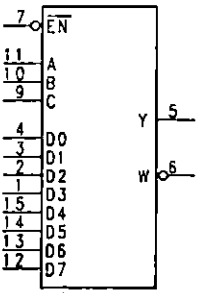
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DE	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 86 OF 113



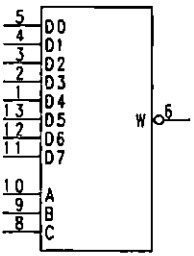
\148



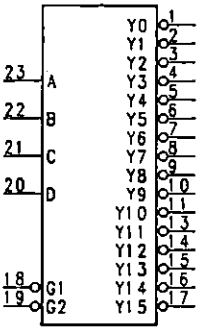
\150



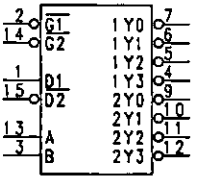
\151



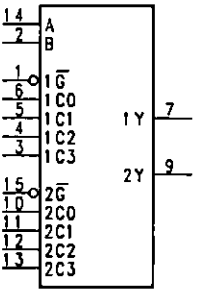
\152



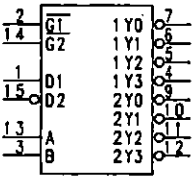
\154



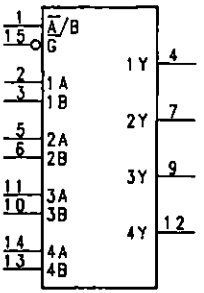
\155



\153

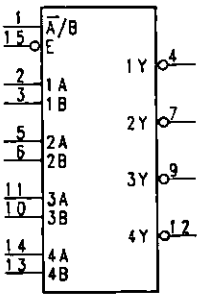


\156

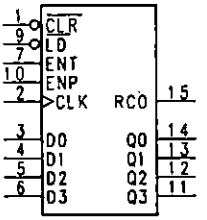


\157

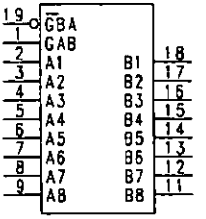
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 87 OF 113



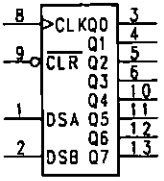
\158



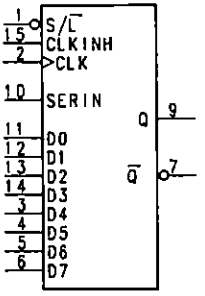
\160



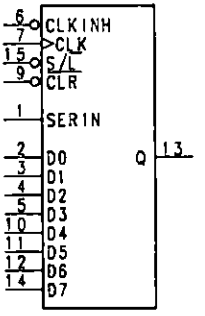
\1620



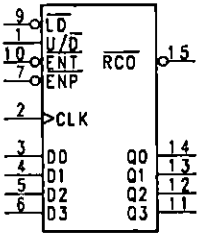
\164



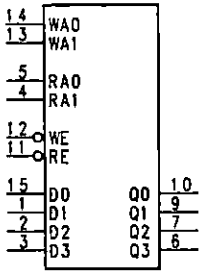
\165



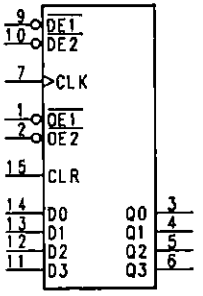
\166



\168

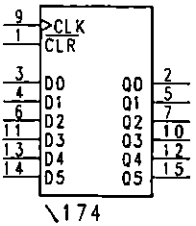


\170

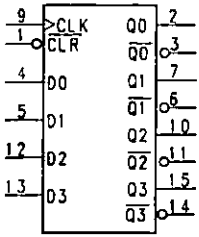


\173

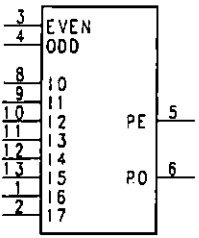
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 88 OF 113



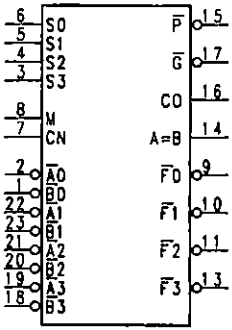
\174



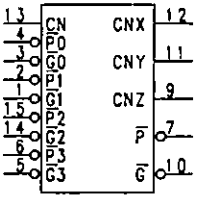
\175



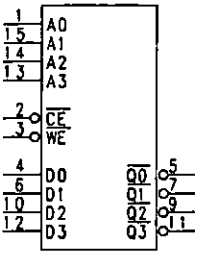
74180



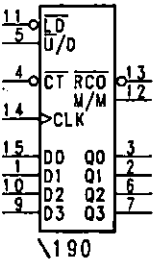
\181



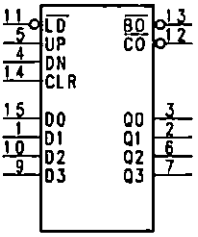
\182



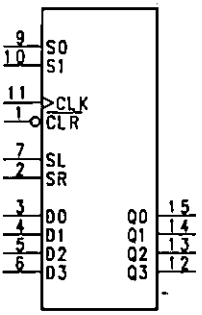
\189



\190

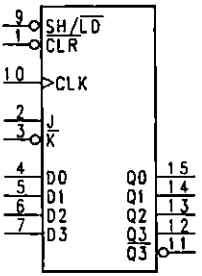


\192

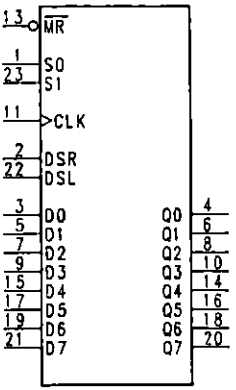


\194

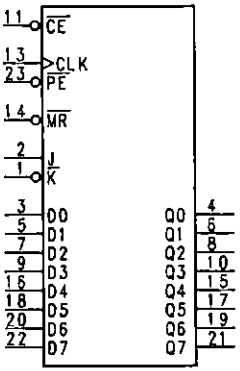
DRAWN BY: L.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DD	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 89 OF 113



\195

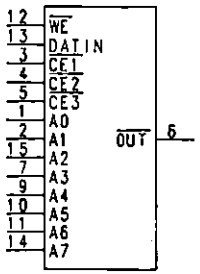


\198

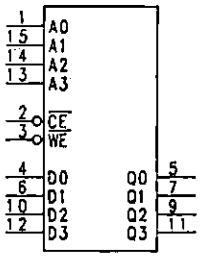


\199

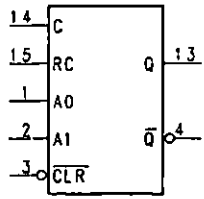
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 90 OF 113



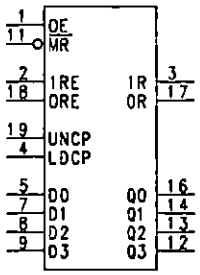
\200



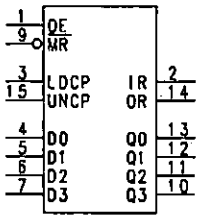
\219A



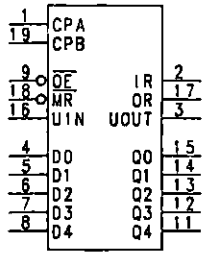
\221



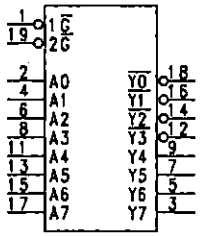
\222



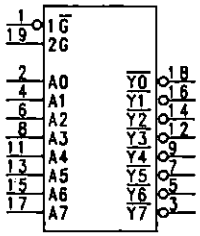
\224



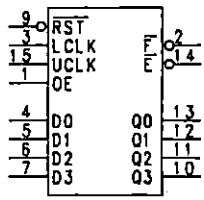
\225



\230

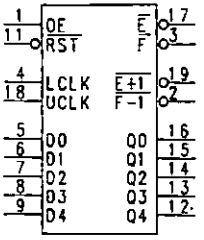


\231

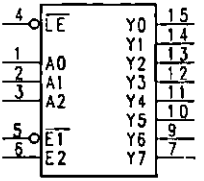


\232

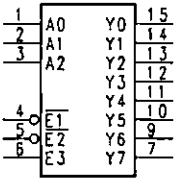
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1.3.90			
APPROVED BY:	DATE	TITLE 54/74 SERIES (\ XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 91 OF 113



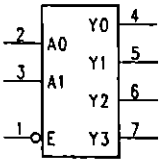
\233A



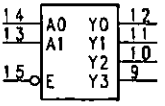
\237



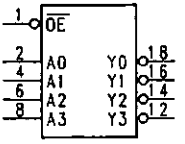
\238



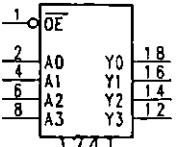
\239



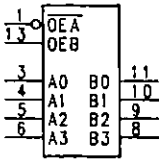
\239S



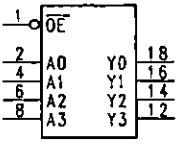
\240



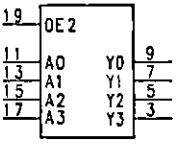
\241



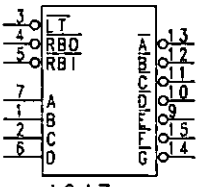
\242



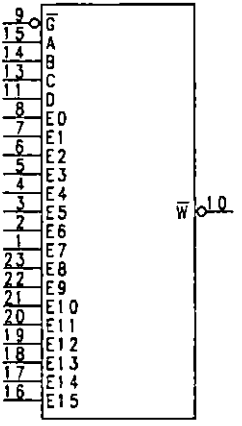
\244



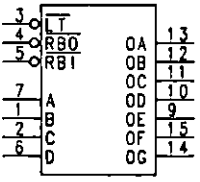
\244A



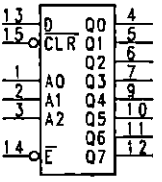
\247



\250

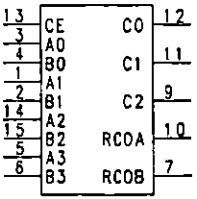


\248

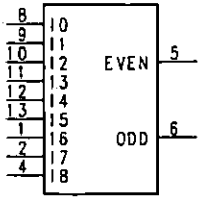


\259

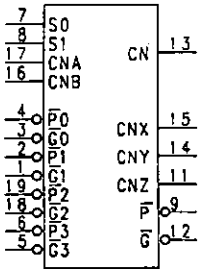
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY:	DATE	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 92 OF 113



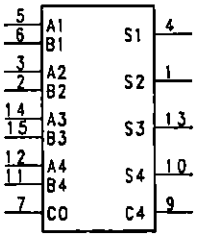
\264



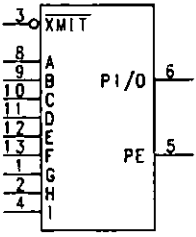
\280



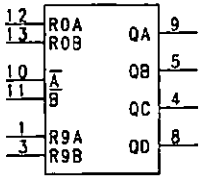
\282



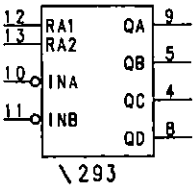
\283



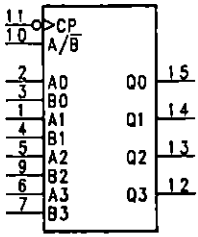
\286



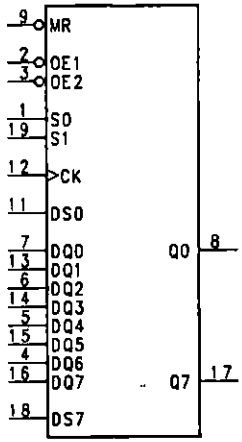
\290



\293

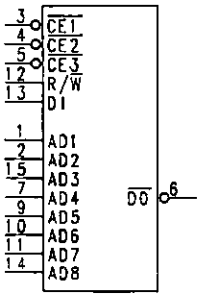


\298

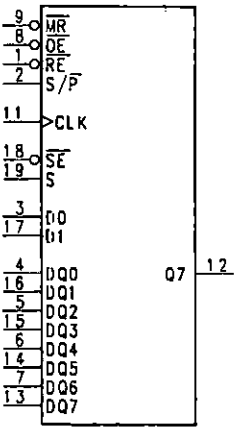


\299

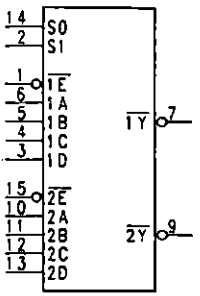
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 93 OF 113



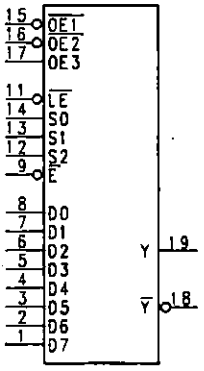
\301



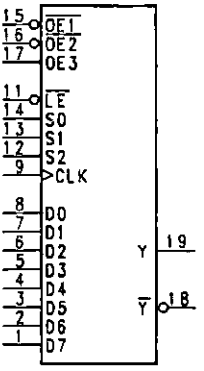
\322



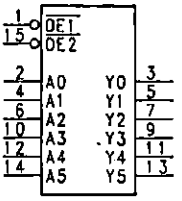
\352



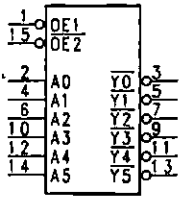
\354



\356

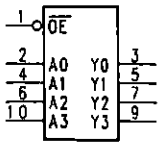


\365

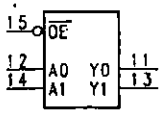


\366

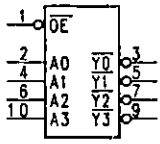
DRAWN BY: J P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: @	DATE 1-3-90			
APPROVED BY:	DATE	TITLE 54/74 SERIES (XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 94 OF 113



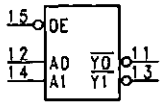
\367A



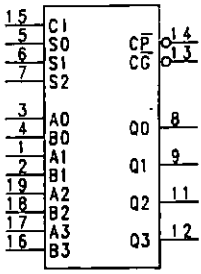
\367B



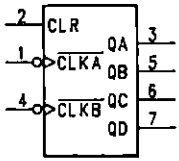
\368A



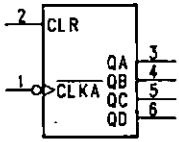
\368B



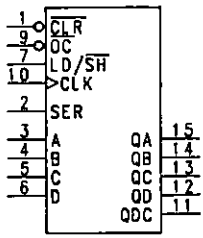
\381



\390

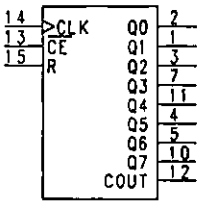


\393

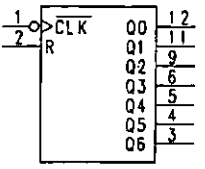


\395

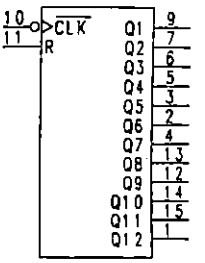
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE	TITLE 54/74 SERIES (XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 95 OF 113



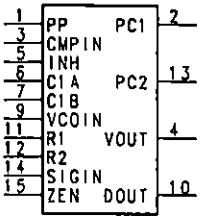
\4022



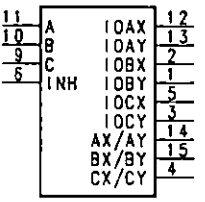
\4024



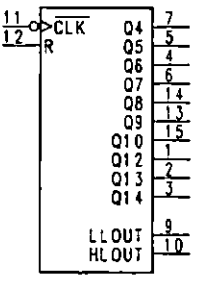
\4040



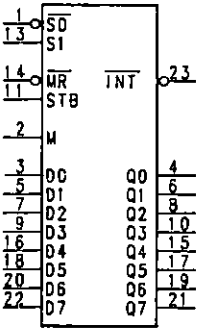
\4046



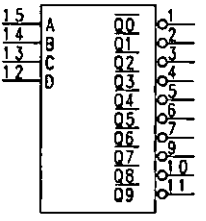
\4053



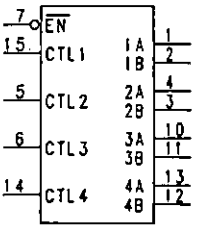
\4060



\412

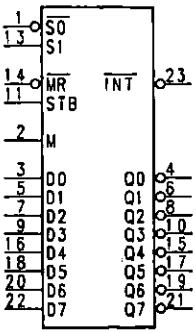


\42

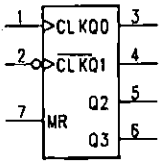


\4316

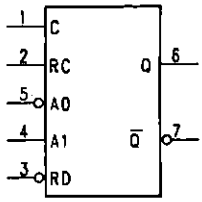
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: @	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 96 OF 113



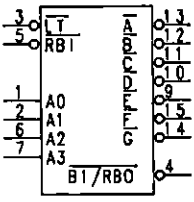
\432



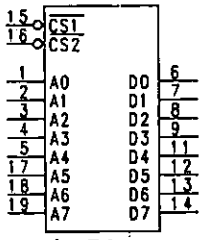
\4518



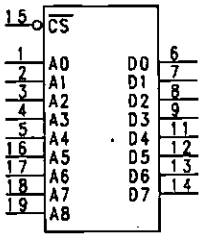
\4538



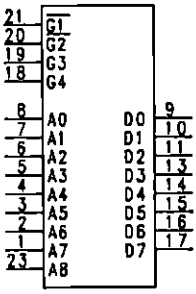
\47



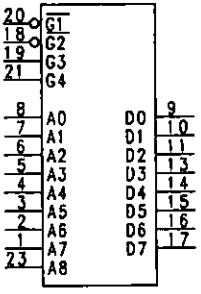
\470



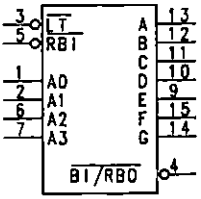
\472



\474

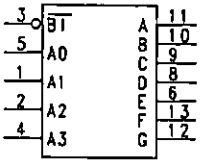


\475

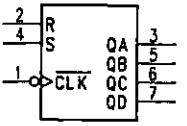


\48

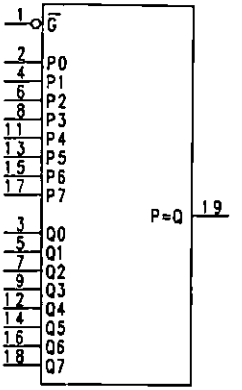
DRAWN BY: J P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DC	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (\ XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 97 OF 113



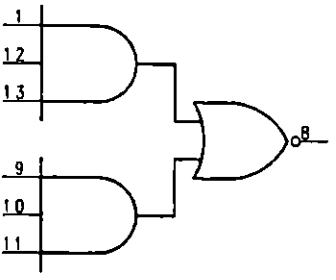
\49



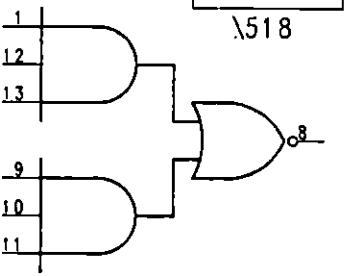
\490



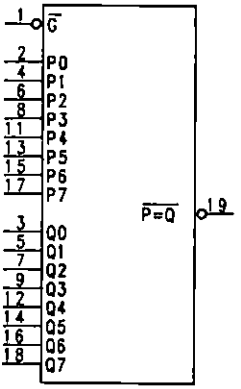
\518



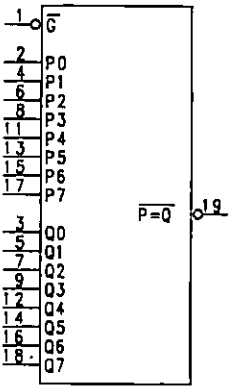
\51A



\51B

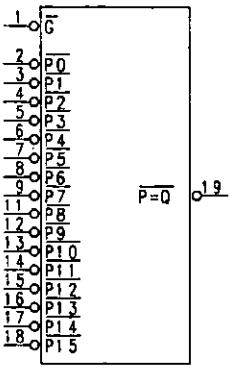


\520

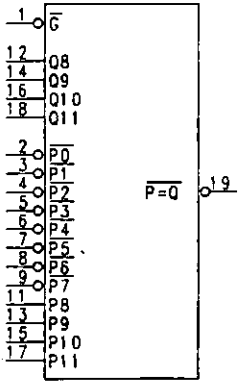


\521

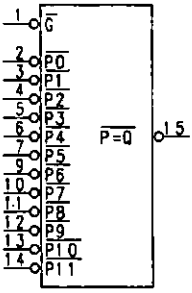
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 98 OF 113



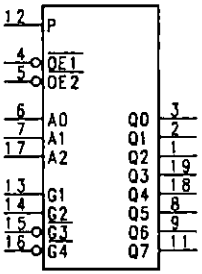
\526



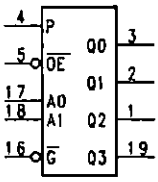
\527



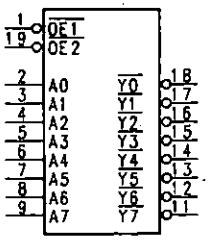
\528



\538

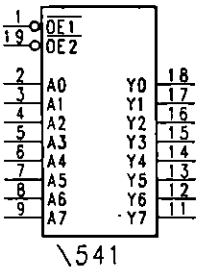


\539

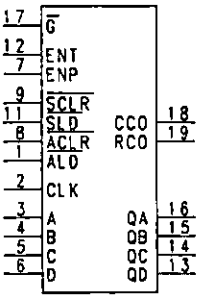


\540

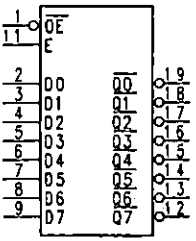
DRAWN BY: J. P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DD	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 99 OF 113



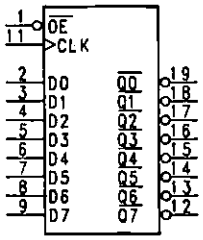
\541



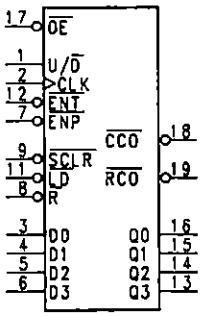
\560A



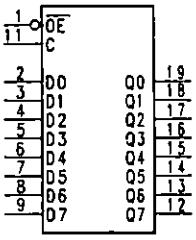
\563



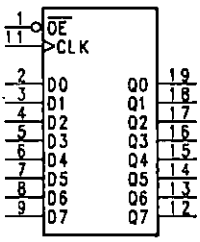
\564



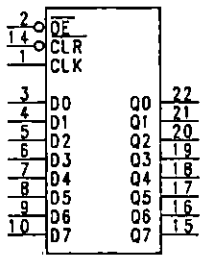
\568



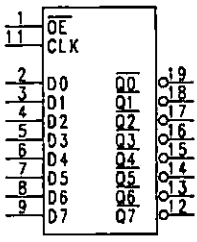
\573



\574

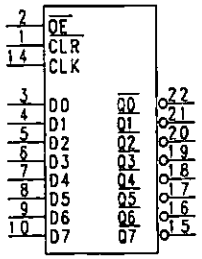


\575

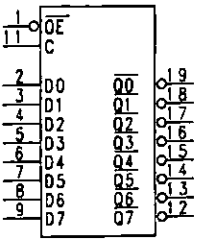


\576

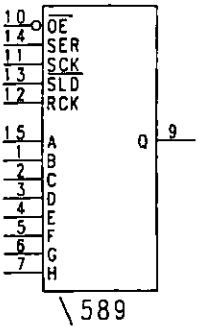
DRAWN BY: J.P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY: (illegible)	DATE 1-3-90		
APPROVED BY: (illegible)	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)	
REVISION:	DATE		
JOB NAME:		REV. A	SCALE 1:1	SHEET 100 OF 113



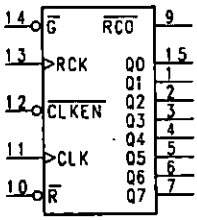
\577



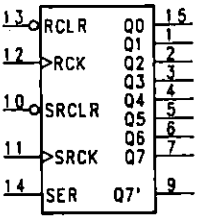
\580



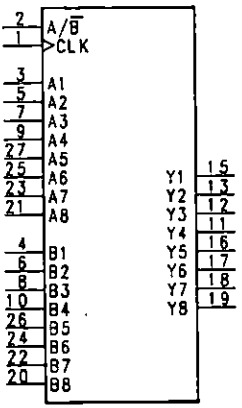
\589



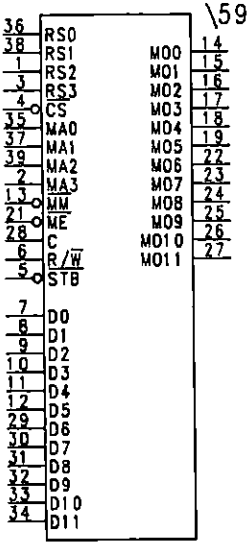
\590



\594

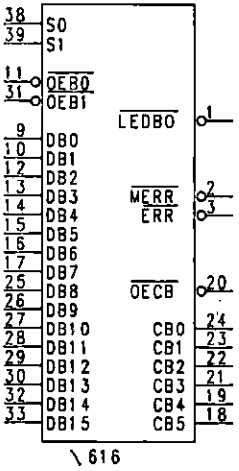


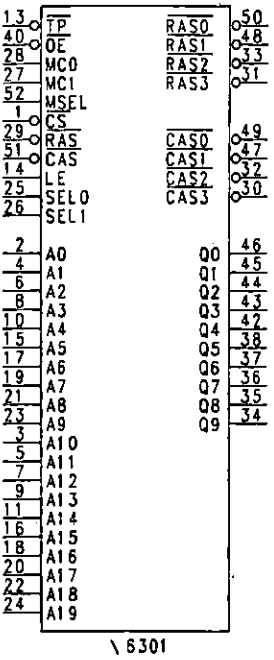
\604

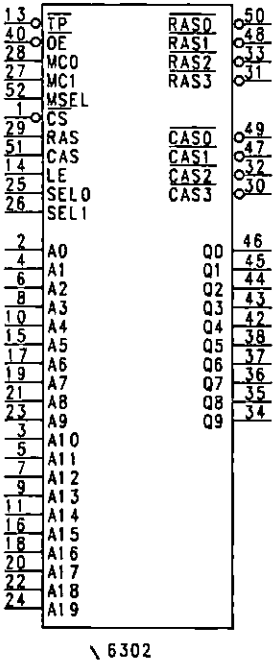


\610

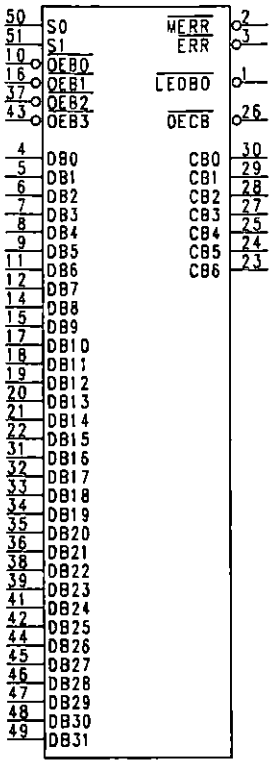
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: DD	DATE 1-3-90			
APPROVED BY: RAB	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 101 OF 113

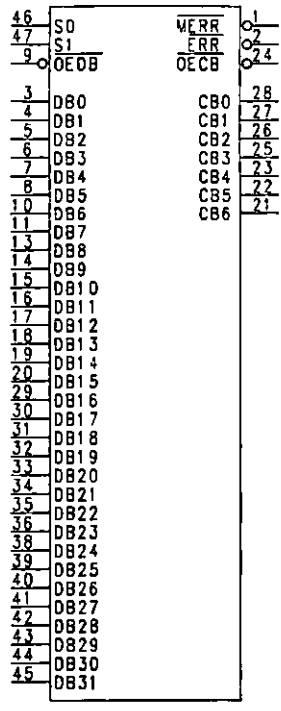


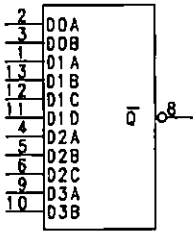


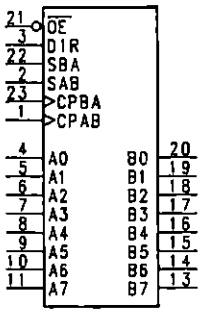


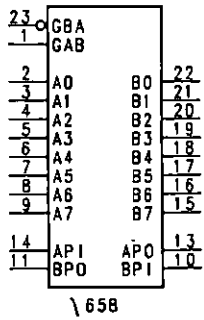
DRAWN BY: J. P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 64/74 SERIES (XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 102 OF 113



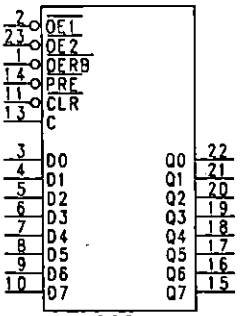


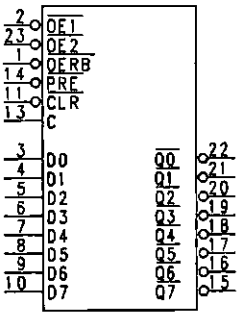


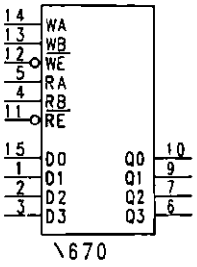


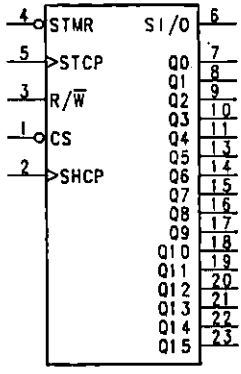


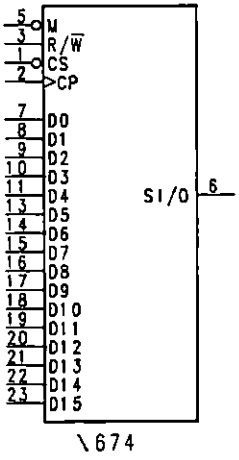
DRAWN BY: RP.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 103 OF 113

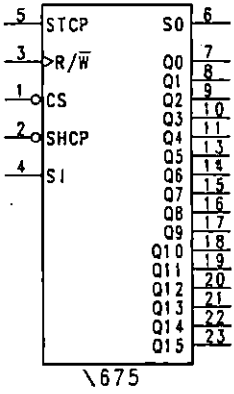




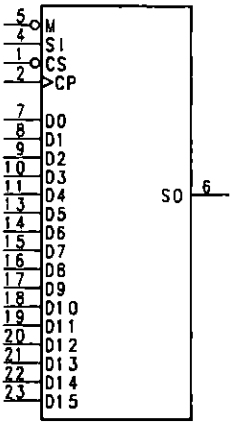




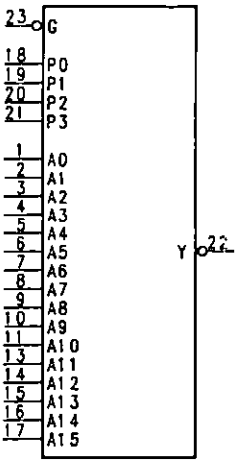




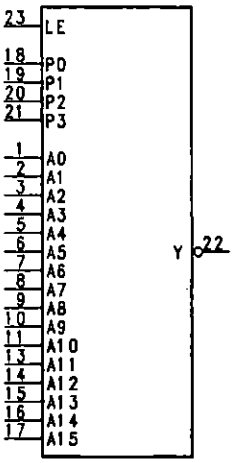
DRAWN BY: [illegible]	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 104 OF 113



\676

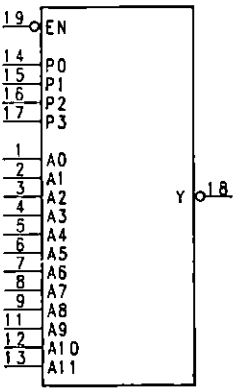


\677

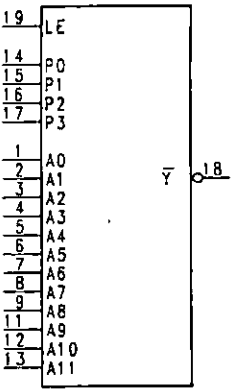


\678

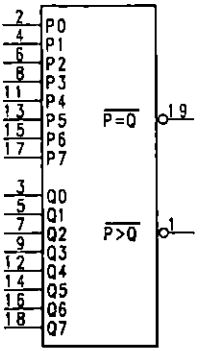
DRAWN BY: J.P	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY:	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 105 OF 113



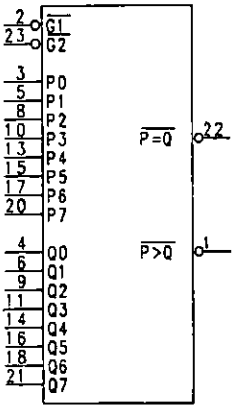
\679



\680

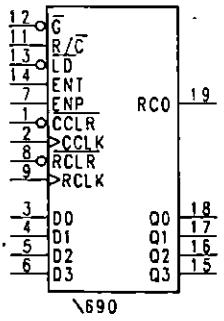


\682

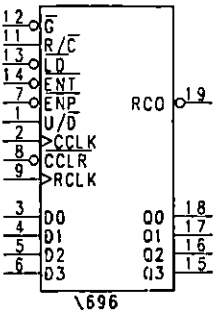


\686

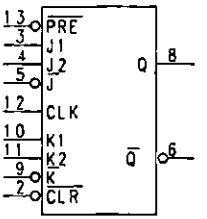
DRAWN BY: J. P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 106 OF 113



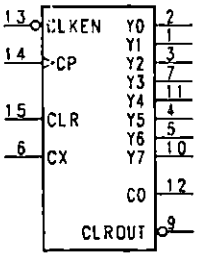
\690



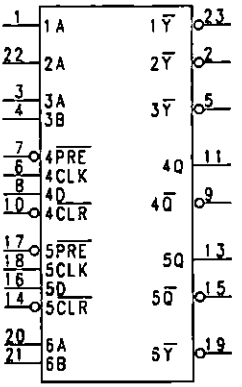
\696



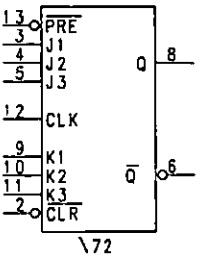
\70



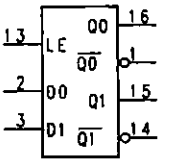
\7022



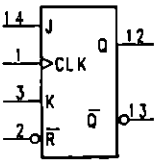
\7074



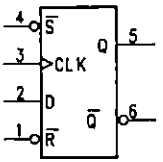
\72



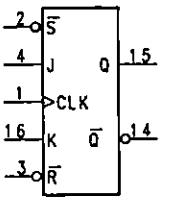
\75



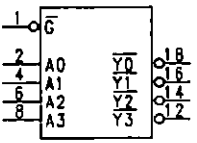
\73



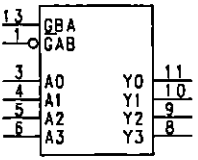
\74



\76

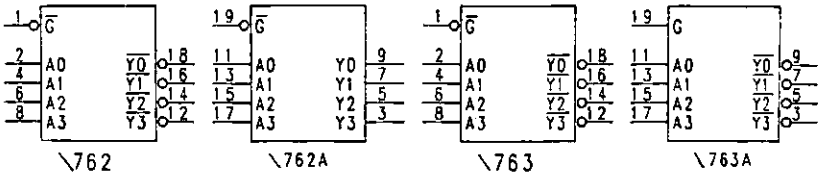


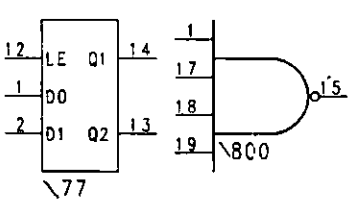
\756

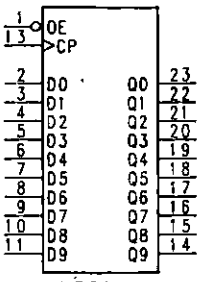


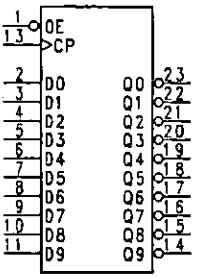
\758

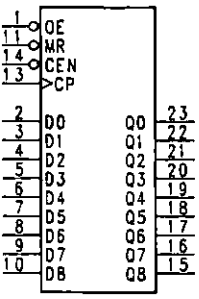
DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 107 OF 113

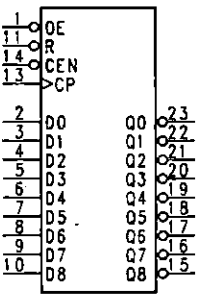


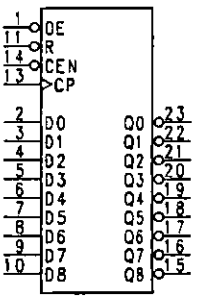


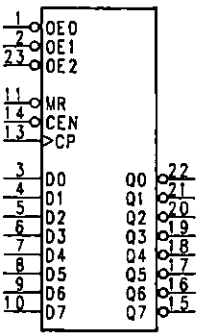


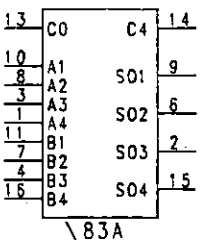




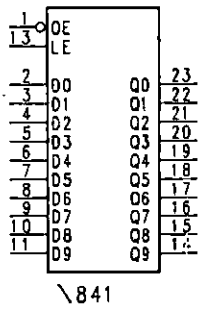




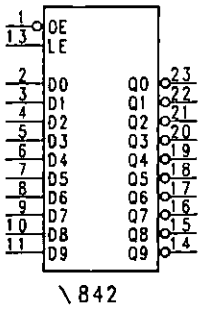




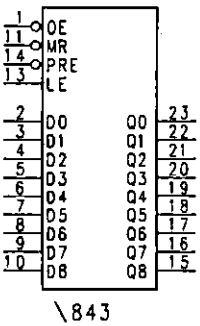
DRAWN BY: KP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE	54/74 SERIES (\XXX)	
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 108 OF 113



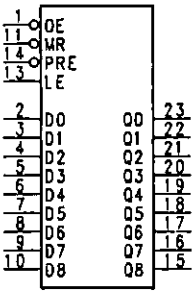
\841



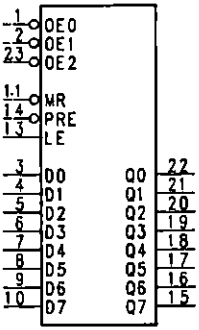
\842



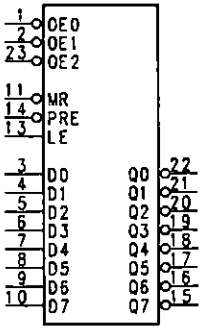
\843



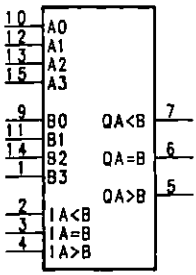
\844



\845

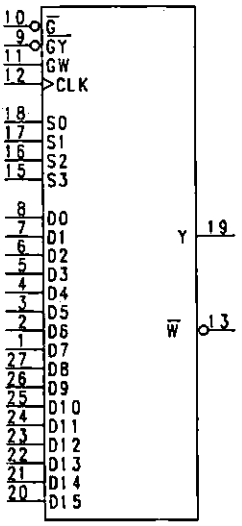


\846

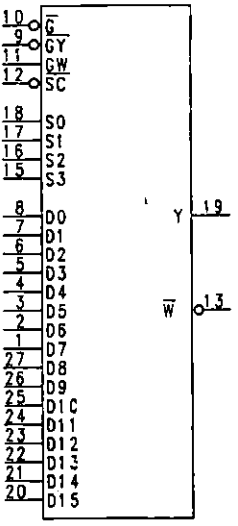


\85

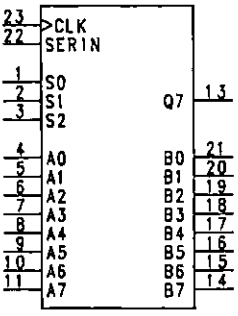
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 109 OF 113



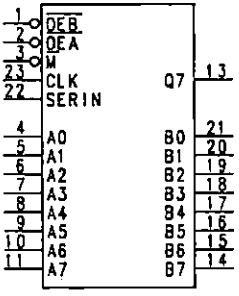
\850



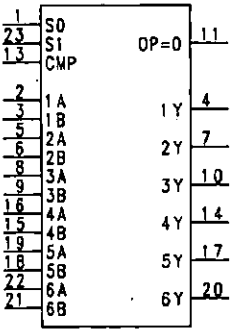
\851



\852

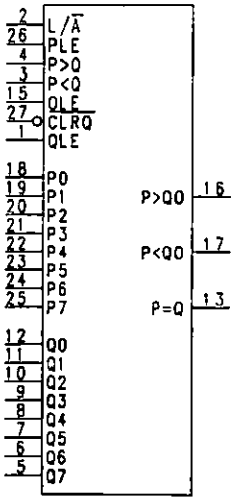


\856

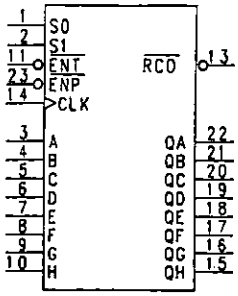


\857

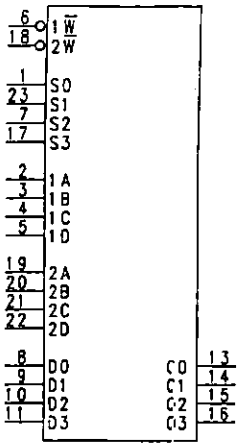
DRAWN BY: J P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: (RD)	DATE 1-3-90			
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 110 OF 113



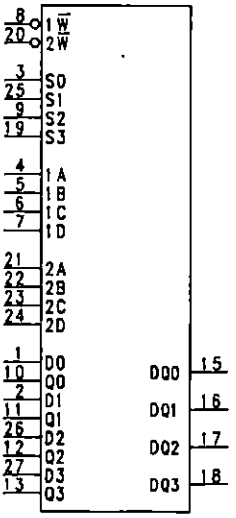
\866



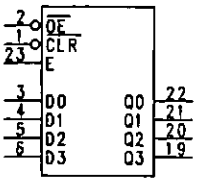
867



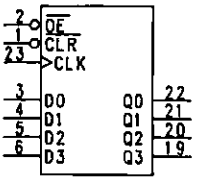
\870



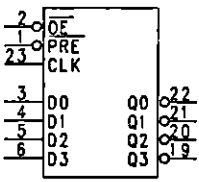
\871



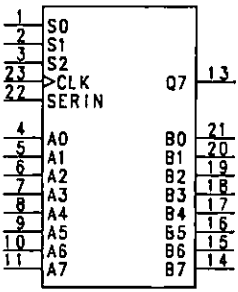
\873



\874

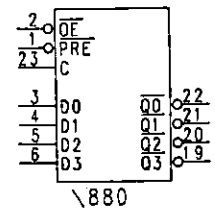


\876

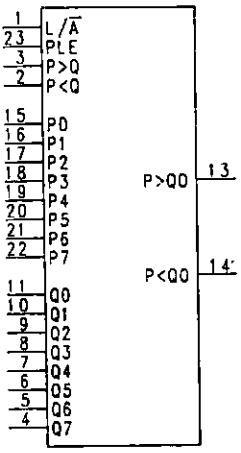


\877

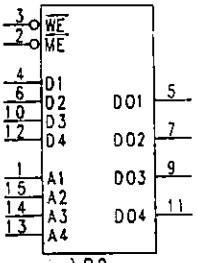
DRAWN BY: JP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460			
CHECKED BY: [illegible]	DATE 1-3-90				
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (\ XXX)			
REVISION:	DATE				
JOB NAME:		REV. A	SCALE 1:1	SHEET 111 OF 113	



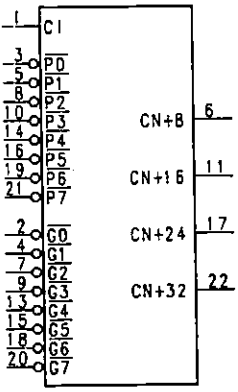
\880



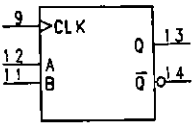
\885



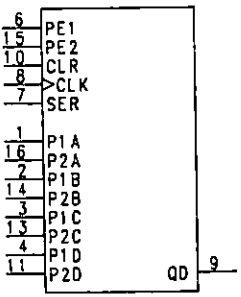
\89



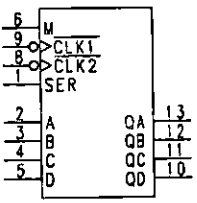
\882



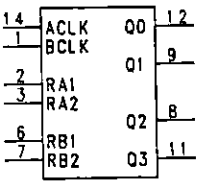
\91



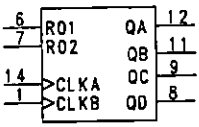
\94



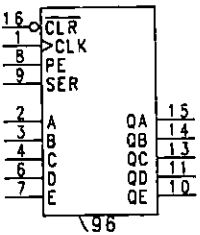
\95



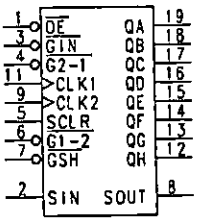
90



\92

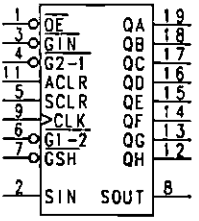


\96

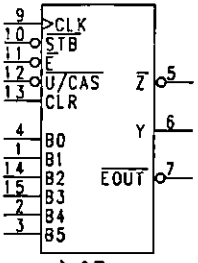


\963

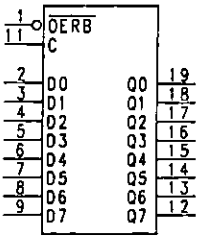
DRAWN BY: J P.	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460	
CHECKED BY:	DATE 1-3-90		
APPROVED BY:	DATE 1-3-90	TITLE 54/74 SERIES (\ XXX)	
REVISION:	DATE		
JOB NAME:	REV. A	SCALE 1:1	SHEET 112 OF 113



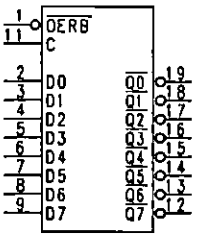
\964



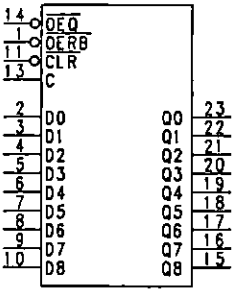
\97



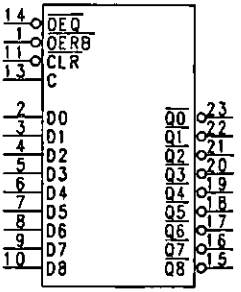
\990



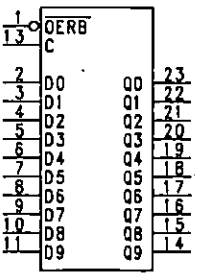
\991



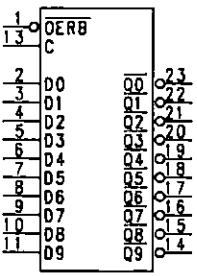
\992



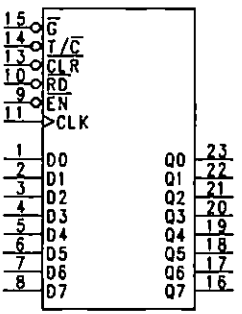
\993



\994



\995



\996

DRAWN BY: LP	DATE 1-3-90	CAD SOFTWARE INC. 119 RUSSELL STREET LITTLETON, MA 01460		
CHECKED BY: [illegible]	DATE 1-3-90			
APPROVED BY: [illegible]	DATE 1-3-90	TITLE 54/74 SERIES (\XXX)		
REVISION:	DATE			
JOB NAME:		REV. A	SCALE 1:1	SHEET 113 OF 113

APPENDIX C

NETWORKING

PADS™-LOGIC

LIBRARY

APPENDIX C

NETWORKING

Users of Multi-stations of **PADS-LOGIC** may want to network all stations together into a PADS Network. For example, it is easier to maintain control over the Library used by all users, if the Parts Library is under the direct control of the Systems Manager. This control is possible when **PADS-LOGIC** is networked, as described in this Appendix.

Should the User wish to network LOGIC, we suggest the use of the NOVELL Network, as produced by Novell, Inc., of 122 East, 1700 South, Provo, Utah 84601.

For illustration purposes, assume a five (5) Terminals PADS Network. Terminals A, B, C & D are Design Stations. Terminal E is both a Design Station and the Systems Manager Terminal where the System Library is maintained. All parts used in Design on any of the Five Design Terminals are to be obtained from the Systems Manager Library on Terminal E. All parts created or modified on any of the Five Design Terminals are to be stored in the Systems Manager Library. To implement such an arrangement, proceed as follows:

1) Implement the network, using NOVELL. Once the network has been implemented, verify files can be transferred between stations. Transfer ASCII and Job files to and from each station. When satisfied these files are transferrable, proceed with Step 2.

2) Set all library files as shareable READ/WRITE. Use the NETWORK utility "flag" to examine and set the files. In the example shown below, "X:>" equials the network drive.

To set up the library files, proceed as follows:

```
X:> CD  \PADS\LIB
X:\PADS\LIB>FLAG   *.IDX SRW
X:\PADS\LIB>FLAG   *.DAT SRW
```

To examine the library files, type the command: FLAG

X:\PADS\LIB>FLAG <CR>

SPART.DAT SHAREABLE READ WRITE
SPART.IDX SHAREABLE READ WRITE
ETC.,.
(where FILENAME NETWORK_ATTRIBUTE)

All systems on the network must run "SHARE" (share.exe), this is part of MS-DOS.

c:>SHARE

3) On terminals A, B, C, and D, (But not E, the Systems Manager Terminal), edit the library manager files PLIB.STD and PLIB.USR to reflect the System Manager terminal. These edits are changes to the Terminal directory path. By modifying these paths, terminals A, B, C, and D will obtain library parts from the System Manager's library on Terminal E. To edit these files, proceed as follows:

Be in the root Directory with C: > displayed at the CRT, using the command:

PEDIT \PADS\LIB\PLIB.USR <cr>

The PLIB.USR file should now be displayed on your CRT:

```
4444
0    \ PADS\LIB\ULINE.DAT        XXX  etc.
  4  \ PADS\LIB\ULINE.1DX        XXX  etc.
    0 20 0

1    \ PADS\LIB\UPCBDEC.DAT      XXX  etc.
  5  \ PADS\LIB\UPCBDEC.1DX      XXX  etc.
   0 17 0

2    \ PADS\LIB\UCAEDEC.DAT      XXX  etc.
  6  \ PADS\LIB\UCAEDEC.IDX      XXX  etc.
  0 17 0
```

```
3   \ PADS\LIB\UPART.DAT        XXX  etc.
 7  \ PADS\LIB\UPART.IDX        XXX  etc.
  0 17 0
```

Note that there are eight lines of data in the file, two lines of data for 2-D Lines, two lines of data for Decals, two Lines of data for CAE Decals, and two lines of data for Part Type data.

Using the PEDIT Editor, place an E: at the beginning of EACH of EIGHT lines of data, such that they appear as follows:

```
0   E: \ PADS\LIB\ULINE.DAT     XXX  etc.
  4 E: \ PADS\LIB\ULINE.IDX     XXX  etc.
    0 20 0

1   E: \ PADS\LIB\UPCBDEC.DAT       XXX etc.
  S E: \ PADS\LIB\UPCBDEC.IDX

    etc.      etc.
    etc.      etc.
    etc.
```

After this is done, save the file by overwriting the existing file.

Next, change the standard PLIB.STD file, by using the PEDIT Editor and the command.

PEDIT \ PADS\LIB\PLIB.STD <cr>

Call up the PLIB.STD file. It will appear, as follows:

```
0  \ PADS\LIB\SLINE.DAT         etc.  etc.
  4 \ PADS\LIB\SLINE.IDX         etc.  etc.
    0 20 0

1   \ PADS\LIB\SPCBDEC.DAT      etc. etc.
  5 \ PADS\LIB\SPCBDEC.IDX      etc.
    0 17 0

    etc.
    etc.
```

etc.
etc.

for eight data lines.

Edit the file by placing an E: at the beginning of each of the eight lines of data, as in the User File above, and save the file by overwriting the Existing file.

3) Repeat the above Editing on terminal A, B, C & D. Test each Terminal Station to insure PADS created parts at the station are stored on Terminal E, the System Manager's Library, and can be retrieved from that Terminal.

NOTES:

(1) While the NOVELL Network is recommended, do not use the NOVELL ARCNET card if planning to use PADS-SUPER ROUTER.

(2) Users should avoid performing simultaneous writes or updates to the library.

(3) Library inititalization should not be attempted across the network, and should only be attempted on the local system.

PADS™-LOGIC

USER MANUAL

INDEX A-Z

INDEX

Notes:
Roman Numerals in ( ), refer to PART of Manual, i.e., Part I, II or III.
1.2 refers to Chapter 1, paragraph 1.2.
1-2 refers to Chapter 1, page 2.

A

Add Bus	(II) 4
Add Connections	(II) 1.2; (III) 3.1; (III) 4.0
Add Gnd	(III) 4.5
Add Lines	(II) 5.1
Add Parts	(II) 1.2; (II) 2.0; (II) 2.1; (III) 3.1
Add Power	(III) 4.6
Add Text	(II) 5.2; (III) Chapter 10
Alternate Part	(II) 2.2, (III) 14-8
Archival Files	(II) 10.6
ASCII Files	(III) 2.3.1
ASCII IN	(III) 2.4
ASCII OUT	(III) 2.5
Auto Pan	(II) 1.7.4; (III) 1.5; (III) 8.3.5

B

Backup Internal	(III) 8.3.3
Bill of Material	(III) 2.8.2
Binary Files	(III) 2.1
Bus	(II) 4
Bus, Adding Connections	(II) 4.2
Bus Angle	(III) 8.3.7
Bus, Grouping	(II) 4.3
Bus Line Width	(III) 8.3.6

C

Combine(Text/Lines)	(II) 5.1.7; (II) 5.2.
Command Menu	(II) 1.4
Connections, Add	(II) 3.1
Connections, Delete	(II) 3.7
Connections, Move	(II) 3.5
Connections, Rename	(II) 3.6
Connections, Setup	(II) 3.8
Connections, Width	(III) 8.3.8
Contract	(II) 1.7.6
Copy	(II) 1.2
Copy Lines	(II) 5.1.4; (III) 6.4
Copy Part	(II) 2.3; (III) 6.1
Copy Text	(II) 5.2.1; (III) 6.3
CRT:	
Initial Presentation	(II) 1.2
Features	(II) 1.3; (III) 1.4
Manipulation	(II) 1.7; (III) 1.5

D

Decal, Alternate PCB	(III) 14-24
Delete Bus	(II) 4.4
Delete Files	(II) 10.4
Delete Net	(II) 3.7
Delete Part	(II) 2.6
Delete Routine	(III) 7.2
Delete Text	(II) 5.2.3
Demo Schematic	Fig. 1A, B, C and D
DeMorgan Equivalents	(II) 2.2
Directories	(I) 1.5; (II) 10.2

E

ECO,:	
To PCB	(III) 2.7.1
From PCB	(III) 2.7.2
Edit Part Type	(III) 5.1.2
Edit Part Text	(II) 2.7
Edit Text	(III) 5.1

Edit Text Ref Des (III) 5.1.1
Eng. Change Order (III) 2.7
Expand (II) 1.7.5

G

Gate Decal (III) 14.5
Global,:
Part Text Size (II) 2.5
Grid Dot (II) 1.6; (III) 8.3.1
Grid System (II) 1.6
Ground Symbol (III) 14.3.2
Group,:
Bus (II) 4.3; (II) 6.1.6
Copy (II) 6.1.2
Create (II) 6.1; (III) 12.1
Cut (II) 6.1.5
Delete (II) 6.1.3
Hierarchical (III) 12.5
Move (II) 6.1.1

H

Hardware Required (I) 1.7
Hierarchy (II) 8.0; (III) Chapter 15
Hierarchical Background (III) Chapter 12
Hiearchical Symbol, Menu (III) 15.1
Hiearchical Modify (III) 15.2
Hiearchical IN/OUT (III) 15.4

I

Input/Output Routine (III) Chapter 2
Installing Software (I) 1.8

J

Job In (III) 2.2
Job Out (III) 2.3

L

Library	(II) 9.0
Library,:	
Creating A Connector	(II) 9.3; (III) Chapters 13, 14, 14.2
Creating Multi-Gate Part	(II) 9.1; (III) Chapters 13, 14
Creating Single Gate Part	(II) 9.2; (III) Chapters 13, 14
Electrical Data	(III) 13.1.1; (III) 14.1.1, 14.2.1
Gate Data	(III) 13.1.2, 14.2.2
Maintenance	(III) 13.10.1
Off Page	(III) 14.3
Standard	(III) 13.10.2.2
Lines,:	
2D	(III) Chapter 11
Arcs, Circle	(III) Chapter 11
Combined	(III) Chapter 11.4
Real Width	(II) 5.1; (III) 8.3.2
Loading Software	(I) 1.8

M

Macros,:	
Auto	(II) 7.5; (III) 16.5
Create	(II) 7.4; (III) 16.2
Commands	(II) 7.2; (III) 16.11
Delete	(III) 16.2.3
Language	(II) 7.8.2
Macro Routine	(II) 7.0
Menu Macros	(II) 1.7.10; (III) 1.4.7
Menu, Selection	(II) 1.5; (III) 1.4.7
Mirror Part	(II) 2.3
Move:	
2-D Lines	(II) 5.1.
Bus	(II) 4.1
Connection	(II) 3.6; (III) 7.1.2
Part	(II) 2.4
Part Text	(II) 2.5
Pin Numbers	(II) 2.5
Text	(II) 2.5

N

Net Delete (II) 3.7
Net Name (II) 3.6
Net Name Size (III) 8.3.10
Net Rename (II)3.6
Next Type (II) 2.3

O

Off Page Symbol (II) 3.4; (III) 14.3
Off Page Replace (III) 14.4

P

Pan (II) 1.7.3
Pan, Auto (II) 1.7.4; (III) 1-14; (III) 8.3.5
Pan Delay (III) 8.3.5.4
Pan Sensitivity (III) 8.3.5.3
Part, Alternate (II) 2.2
Part, Copy (II) 2.3
Part, Create (See Library)
Part, Delete (II) 2.6; (II) 9.6
Part, Mirror (II) 2.3
Part, Move (II) 2.4
Part, Next Type (II) 2.3
Part, Rotate (II) 2.3
Part, Setup (II) 2.11
Part, Text (II) 2.5
Part Type Edit (II) 2.10
Parameters, Setup (III) 8.3
Pin, Numbers (II) 2.5; (II) 2.9
Pin, Number Swap (II) 2.9;(III) 5.1.3
PKGTYP.DAT File (II) 9.5; (III) 14.7
PLIB (III) 13.2
Power Symbol (III) 14.3.3

R

Real Width	(II) 5.1; (III) 8.3.2
Reference Designators	(II) 2.8
Registration, Warranty	(I) 1.9
Repaint	(II) 1.7.1; (III) 1-14
Repaint, Stop	(II) 1,7,8; (III) 1-15
Reports	(III) 2.8
Reports:	
Board	(III) 2.8.2
Limits	(III) 2.8.6
Net List	(III) 2.8.1
Off Page	(III) 2.8.3
Status	(III) 2.8.4
Unused	(III) 2.8.5
Reset	(II) 1.7.2
Rotate Part	(II) 2.3

S

Search Macro	(II) 2.12
Setup	(II) 1.2; (III) Chapter 8
Sheets	(III) 2.6.1
Sheets,:	
Add	(III) 2.6.2
CRT	(III) 8.2
Delete	(III) 2.6.4
Macro	(III) 8.4
Parameters	(III) 8.3
Rename	(III) 2.6.3
Routine	(III) Chapter 8
Select	(III) 2.6.1
Size	(III) 8.3.4
Shell (PADS)	(I) 1.5; (III) 1.2

T

Text:
Add (II) 5.2
Copy (II) 5.2.1
Delete (II) 5.2.3
Edit (II) 2.7; (II) 5.2.2
Justification (III) Chapter 10
Move (II) 5.2.4
Rotate (III) Chapter 10
Size (II) 2.5; (II) 5.2
Thickness (II) 2.5; (II) 5.2
Tie Dot (III) 8.3.9
Tolerance, Part (III) 5.1.4

U

Unique Features (I) 1.3

V

Value, Part (III) 5.1.4

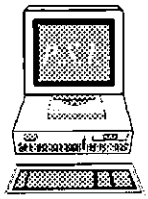
W

Warranty (I) 1.9
Working Area (III) 1.4.2

PADS®-LOGIC

Version 2.5 Update Guide

- New Features
- The 32-Bit Option
- Installation Notes
- Review of Operational Basics



PADS-Logic V.2.5
Update Guide
©PADS Software Inc. 1992

All rights reserved. No part of this publication may be reproduced, transmitted, transcribed, stored in a retrieval system, or translated into any language in any form by any means without the written permission of PADS Software, Inc.

Contents

Introduction 1
Hardware Requirements 1
Installing PADS-Logic 2
Other Considerations 3
Graphics Device 4
Mouse Driver 4
Directories Created By PADS-Logic 4
Installation Procedure 4
Starting PADS-Logic 10
The Graphical User
    Interface (GUI) 10
Menu Organization And
    Command Selection 13
Mouse Functionality 14
Mouse Speed Control 15
Theory Of Operation 16
Technical Support 18

INTRODUCTION

This guide is provided as an update supplement to the PADS-Logic reference manual. Please pay particular attention to the details concerning the new, 32-bit option before installing the new release.

This guidebook covers:

- the hardware requirements for the two PADS-Logic options
- how to install PADS-Logic V2.5
- starting PADS-Logic V2.5
- the PADS-Logic graphical interfaces
- a review of PADS-Logic's newer operational procedures.

HARDWARE REQUIREMENTS

PADS-Logic can now be operated in either of two modes; 16-bit or 32-bit. If you are operating PADS-Logic on a 80286 based computer, you should select the 16-bit option during the installation. If you will be operating PADS-Logic on a 80386 or 80486 based computer, select the 32-bit option. The 32-bit option takes advantage of the enhanced architecture of 32-bit microprocessors.

For the 16-bit option, the hardware requirements are:

- IBM-AT, 80286, 80386 or 80486 computer.
- 640K DOS Memory, plus a minimum of 640K of expanded memory. For best performance, we recommend a total of 1MB of expanded memory.
- If you are using an AT-286 computer, you may have to add an Expanded Memory board. If you have a 80386/486 computer, the expanded memory may be defined using expanded memory management software.

For the 32-bit option, the hardware requirements are:

- 80386SX, 80386, 80486 or compatible computer.
- 640K DOS memory, plus 2MB of additional extended memory.

Both the 16-bit and the 32-bit options of PADS-Logic require a:

- 2-button or 3-button mouse (A 3-button mouse is recommended).
- communications port for the mouse.
- EGA, VGA, or other approved color graphics adapter and compatible color monitor.
- Floppy disk drive. (For software installation only)
- Minimum 20 MB hard disk drive.

Optional equipment includes:

- Pen plotter, matrix printer, or laser printer output device.
- Communications port for the output device.

A complete list of supported graphic cards, pen-plotters, laser printers, and matrix printers is presented when you install PADS-Logic with the PINSTALL program.

INSTALLING PADS-LOGIC

Both options of PADS-Logic are similar in operation, and compatible with all of our PCB design programs. Before beginning the installation process, decide which option you will be installing. To help you select which option to install a list of considerations is provided below.

Other considerations

PADS-PCB does not have the same database limits as the 32-bit option of PADS-Logic. The 32-bit option can be used with PADS-PCB. However, caution must be taken that a schematic is not created that would exceed the limitations of your PCB database limits.

Database limits with the 32-bit option are much larger than the 16-bit option-- over 5100 parts compared to 1250 parts for the 16-bit. All of the limits per sheet are approximately 4 times higher with the 32-bit option.

The maximum number of pins in a part type with the 16-bit option is 255. With the 32-bit option, the maximum number is 999.

Net names for the 16-bit option are limited to 12 characters, same as PADS-PCB.

Net names for the 32-bit option can be 47 characters, same as PADS-2000 (version 3.0 or greater).

A schematic binary file created with the 16-bit option is upward-compatible with the 32-bit option.

A schematic created with the 32-bit option must be transferred to the 16-bit option using an Ascii file transfer method. Keep in mind that the 32-bit database must stay within the limits of the 16-bit option to transfer properly.

The 16-bit option requires expanded memory similar to PADS-PCB, and it has the same Graphical User Interface (GUI) as PADS-PCB.

The 32-bit option requires extended memory (2 MB minimum, 4MB preferred) similar to PADS-2000. It has the same GUI as PADS-2000, and runs only on 80386 (or higher) based computers.

The 32-bit option operates at a faster speed than the 16-bit option.

Both options are compatible with the standard PADS part libraries.

Once you have decided which option of PADS-Logic to install, you may begin the installation procedure.

Graphics Device

Determine the manufacturer and model of the graphics adapter that is installed in your computer. There are numerous selections within the installation program, and it would be best to choose the one that is 100% compatible with your graphics adapter.

Mouse Driver

PADS does not provide a driver for your mouse. The mouse driver must be installed and functioning prior to starting PADS-Logic. To install the mouse driver, consult the instruction manual that you received with your mouse.

Directories Created by PADS-Logic

PADS-Logic may be installed on any drive on your hard disk. When the software is installed, the following directories are created on the disk drive:

Directory Path	Files	Disk Size
<drive:>\PADS	Logic programs	2.5 MB
<drive:>\PADS\LOGFIL	Schematic/user files	0.5MB
<drive:>\PADS\LIB	Library files	5.5MB

Installation Procedure

Throughout this manual, <CR> is used to indicate that you must press the Enter key.

To install PADS-Logic in a network, consult the Networking PADS Programs appendix in your reference manual.

1) Place the disk labeled PADS-Logic, Disk 1 of 2 in floppy disk drive A. From the DOS prompt, type:

   A:PINSTALL <CR>

2) A pop-up window with three options: SOFTWARE INSTALLATION, HARDWARE SETUP, and WORKSTATION SETUP, is displayed.

   Select SOFTWARE INSTALLATION by moving the highlight bar with the up or down arrow keys and then press function key **F1**.

3) A list of programs to install is presented in the pop-up window. Move the highlight to PADS-LOGIC and press F1.

4) A pop-up window with the two options: PADS-Logic (16-bit), or PADS-Logic (32-bit) is displayed. Move the highlight over the option of your choice and select F1.

5) The hardware requirements for the selected option are displayed. You are asked to specify the hard disk drive on which to install PADS-Logic. Type the appropriate drive name corresponding to where your existing PADS-Logic directory resides.

6) You are next asked to define the floppy disk drive from which PADS-Logic is being installed: A or B.

7) Your selected source and destination drive are displayed and you are asked to confirm the selections. If the choices displayed are correct, respond with **Y**. If incorrect, respond with **N**, and repeat steps 5 and 6.

8) You are prompted:

   OK to load main PADS-Logic software?

   Respond **Y** to initiate the installation process. You are prompted to insert the appropriate disk before you may continue with the installation. Respond **Y** again when you are sure that the correct floppy disk is inserted. A series of status messages tell you that the program is being installed onto the disk drive.

9) After the PADS-Logic program is completely installed, you are prompted:

OK to load training files? (Yes/No)

Respond **Y** to install the training files. These files are used in the tutorial sessions of the training manual.

10) You are next prompted:

OK to load PADS system library? (Yes/No)

If you do not have a PADS library installed, type **Y**.

11) You are prompted to select the location of the library files:

Load the files to A) To software drive B) C)

Select A to install the library into the PADS\LIB directory that was created by the software installation. Options B and C are generally used when installing PADS-Logic on a network.

12) You are then prompted to insert the library disk. Remove the PADS-Logic program disk, insert the library disk, and type **Y** to continue the process.

13) After the library is installed, the message displayed is:

Software Installation has been Completed.

It is Necessary To Run A Hardware Setup.

Respond by selecting F1 to return to the main menu. Move the highlight to HARDWARE SETUP and select F1.

14) Select PADS-Logic, then 16-Bit or 32-Bit, as appropriate.

You are prompted for the disk drive location for PADS-Logic, and the disk drive location of DOS. Confirm the location by selecting **Y**. If incorrect, select **N** and respond with the correct drives.

15) Next, you are asked to select a graphics device driver. PADS-Logic supports a range of EGA and VGA graphics adapters, in addition to specialized high-resolution graphics from Elsa and Metheus. Scroll through the list and highlight the appropriate

graphics driver from the list. If you do not see your specific EGA or VGA graphics card listed, select either option 1. IBM EGA, or 2. IBM VGA. After highlighting the correct driver select F1.

16) The next configuration option is for your matrix printer. Highlight the desired driver to use with your printer. If you do not have a printer, select any option. An unlisted matrix printer can be used if it is compatible with one of the printers listed. To the right of the printer selection is an entry that defines the page width in .001" units. Matrix printer page widths are generally or 8.0" or 13.2". If the current value is not correct, move the cursor to the entry with the right arrow key and enter a new value. After the selection and editing of the entry has been completed, select F1.

17) Next, define the type of laser printer you are using. If you are not using a laser printer, select any entry. Move the highlight bar to the appropriate choice from the list. To the right of the printer selection are entries that define the page width and page length. These are in .001" units, and may be edited by moving the cursor to the entry with the right arrow key and entering a new value. After the selection and editing of the entry has been completed, select F1.

18) Next, select the type of pen plotter you are using from the list of the models supported by PADS-Logic. If you do not have a pen plotter or your plotter is not compatible with any listed, choose any model. Move the highlight bar to the appropriate choice from the list. To the right of the plotter selection there are entries which describe the parameters of the plotter. These may be edited by moving the cursor to the entry and entering a new value.
    - **Pens** is the number of pens supported by the plotter.
    - **X and Y Size** defines the paper size you are using.

- **Pen Width** is the width in mils of the pen. This is used to calculate fill patterns.
- **Rotation** identifies if the paper must be rotated on the plotter to support smaller sheet sizes.

After the selection and editing of the entry has been completed, select F1.

19) The next screen defines the photoplotter type, number of apertures, and output format for photoplotting. Both vector and laser plotters are supported. We recommend that you check with your photoplotting vendor before entering the parameters.
   - **Apertures** defines the number of apertures in the plotter.
   - **Xsize and Ysize** define the plotter table size.
   - **Units** defines the output format, Metric (1) or Inch (0).
   - **Prec** defines the precision of the plotter output. The value is the number of digits to the right of the decimal in the output.
   - **Digits** is number of digits in the output coordinates, including leading or trailing zeros. The decimal point is assumed and does not count as a digit.
   - **Zeros** defines the suppression rules for zeroes in the photoplot file. 0 = No suppression; 1 = Suppress leading zeros; 2 = Suppress trailing zeros.
   - **Coord** defines the coordinate system, 0 = Absolute, 1 = Incremental.
   - **Repeat** defines if a repeated coordinate value for x or y should be suppressed (1) or not suppressed (0) in the Gerber file output. Suppression reduces the size of the output file, but not all plotters are compatible with this.

   After the selection and editing of the entry has been completed, select F1.

20) The Device Status screen lets you define which output devices are on-line, and the communication port that they are attached to. For each printer or plotter, you must define it as on-line or off-line, and then define the port the device uses. The port options are PRN, LPT1-9, and COM1-4. Modify the entries as required and select F1.

21) You are then prompted if you wish to send error messages to a file, or to the printer (printer on LPT1 is the default). Respond **Y** or **N**.

22) You are prompted to update your CONFIG.SYS file to add the statements as displayed. Confirm that you want the program to automatically update your file by selecting **Y.** Entering a **N** will cancel the updating of your CONFIG.SYS file, and you will have to edit the file manually before you attempt to operate PADS-Logic.

23) You are then prompted to update your AUTOEXEC.BAT file to add the statements as displayed. Confirm that you want the program to automatically update your file by selecting **Y.** Entering **N** cancels the updating of your AUTOEXEC.BAT file, and you will have to edit the file manually before you attempt to operate PADS-Logic.

24) This completes the hardware setup. Select <ESC> to exit PINSTALL.

25) Now, if the \PADS directory is not defined in your path statement, edit the statement in your AUTOEXEC.BAT file.

26) Now, if FILES are not set to 40 in your CONFIG.SYS file, edit the statement in your CONFIG.SYS file.

Remove the PADS-Logic and library disks from the floppy disk drive and store them in a safe place. If the AUTOEXEC.BAT or CONFIG.SYS files have been modified, reboot your computer.

STARTING PADS-LOGIC

To run PADS-Logic after installation:

1) Make the \PADS directory your current directory by typing:

   CD \PADS<CR>

2) Type:

   PADS <CR> (16-bit option)
   or
   2000 <CR> (32-bit option)

   to enter the PADS command shell. This shell is used to select from all of the PADS programs.

3) Place the mouse cursor over the box labeled PADS-Logic and click the left mouse button.

4) The PADS-Logic copyright notice and the amount of memory available for the program are displayed. The message:

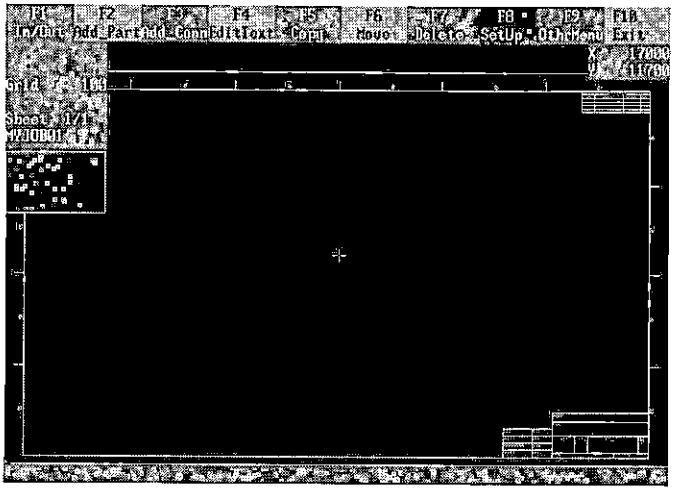
   Press any key to continue

appears. Press any key to start PADS-Logic.

THE GRAPHICAL USER INTERFACE (GUI)

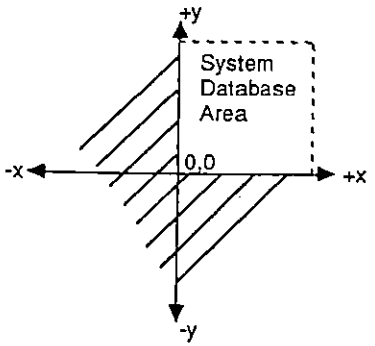
This section introduces the graphical user interface (GUI) and the basic operations of PADS-Logic.

When you start the program, the GUI is loaded, and a D-size drawing sheet is displayed. The 16-bit and 32-bit GUIs are very different. The 16-bit option has menu selections on the left side of the screen, while the 32-bit option places the menus across the top. In the description that follows, the difference between the 32-bit option is followed by the 16-bit in parentheses ( ). The illustrations in this manual have been taken from the 16-bit option. The 32-bit initial screen is displayed below:



The GUI divides the screen into 4 parts: the Working Area, the System Information Windows, the Menu area, and the Prompt Line.

The **Working Area** is the major portion of the graphic screen, currently black, where the D-size drawing is displayed. If you want to use a different sheet size, you may change this with the Setup Parameters command. The Working Area uses the upper right quadrant of a Cartesian coordinate system below:



The system origin is at the lower left corner of the database coordinate system and corresponds to the lower left corner of the screen. All cursor positions within a drawing sheet have positive values. The Working Area has a maximum X and Y size of 64.000".

To aid part type creation and placement a matrix of dots appears in the Working Area. This grid provides a reference to size and distance when working at higher magnification levels. The spacing between dots is user-definable.

The Menu area is at the top of the display. (for the 16-bit option it is on the left side of the display.) The Menu area is used to select the command options of PADS-Logic. Upon entering PADS-Logic you are presented with nine main menus labeled F1 through F9, plus F10 Exit.

The 32-bit option has the function key numbers displayed above the corresponding menu option. Enter the new Modeless command **F<cr>** to toggle the display of the function key numbers off or on.

There are two system information windows in PADS-Logic. The top window contains five lines of information. From the top down, the window displays:

- **Menu Selections:** These 2 lines display the last two menu selections made.
- **Grid XX:** The snap grid in thousands of an inch.
- **Sheet X / Y:** The current sheet, X, and the total number of sheets, Y.
- **Drawing Name:** The name of the schematic currently in memory. (Note: in the 16-bit option the Drawing name, Grid, and Sheet X/Y lines are transposed.)

The second information window, called the "Postage Stamp", locates the Working Area within the drawing sheet. The white outline is the outline of the drawing sheet, and the red area is the area currently displayed in the Working Area.

Whenever you are adding connections or moving part types within the schematic, the postage stamp window will be replaced with 4 lines containing information about the item being manipulated.

The 32-bit option has the capability to toggle the display of the System Information Windows. To toggle the display, press the **Tab** key repeatedly until one or both windows are displayed.

The 32-bit option has a dedicated window for the current cursor position and the relative cursor position. The current cursor position is located at the upper right corner of the display, just below the Menu area. To toggle the display of the current cursor portion on or off enter **Ctrl+L.**

The relative cursor position window is located just under the current cursor position window. To toggle the display of the window on or off enter **Ctrl+PgDn** (Num key 3).

These cursor position windows are displayed above the system information windows in the 16-bit option.

Both options have the Prompt Line below the Working Area. The Prompt Line is the primary means of communication between you and PADS-Logic.

Menu Organization and Command Selection

PADS-Logic uses a hierarchical command menu structure which starts with a main menu and offers a series of sub-menus organized for efficient operation. When a sub-menu is selected from the main menu, the name of that menu is displayed in the System Information Window. The command options associated with the sub-menu will appear in the Menu area, replacing the commands of the main menu. Menu commands are hierarchical, so there may be additional command options beneath a sub-menu command.

Commands are selected by either using function keys F1 through F10 or with the mouse. The numbers next to the commands correspond to the function key numbers. To select commands with the function keys, simply select the function key that corresponds with the command in the menu.

Mouse Functionality

All interaction within the program is performed using the mouse or the keyboard. Since a three-button mouse is recommended, the functionality of a three button mouse will be discussed.

The **LEFT** mouse button is generally used to select items. To select an item with the left mouse button, click the button with the cursor positioned over the item. This method of selecting items can be used within the menu area, pop-up file listings, library browse, etc. For the remainder of this manual whenever a reference is made to selecting an item it is assumed this method is being used.

Note: In the 32-bit option, there is an additional select mode. This mode is entered by selecting the menu option in the menu area and not directly over the item. A "select" menu appears, then you position the cursor over the item and click the **LEFT** mouse button.

The **RIGHT** mouse button is used to abort or exit. Whenever a selection is improperly made, or if you wish to exit to the previous menu, click the right mouse button (There is a safeguard that prevents the user from exiting the program in this manner).

The **MIDDLE** mouse button is used to change the highlighted menu option. The default function of the middle mouse button requires clicking the cursor in the menu area to change the highlighted menu option. While in the working area, the middle button is pressed and held

down while moving the mouse to scroll the highlight through the menu area. The highlight does not wrap around, and un-assigned menu items are not skipped over.

The middle mouse button can be configured to move the highlight by clicking the middle mouse button. The highlight moves to the right one menu option at a time. Upon reaching the last menu item it wraps around to the first menu item. This mode does not require clicking the cursor in the menu area to change the highlighted menu option. The highlight wraps around, and un-assigned menu items are skipped over.

To configure PADS-Logic for the alternate mouse function be at the DOS prompt in the PADS directory and type:

ren cursor cursor.tmp

Then re-enter PADS-Logic and the middle mouse button will perform according to the optional configuration. To reverse the configuration simply reverse the file name order in the above DOS command.

Mouse Speed Control

The cursor is the white cross-hair that follows the movement of your mouse. It defines the current location for action in PADS-Logic. The cursor coordinate is displayed in a System Information Window. The cursor on the screen is moved by moving your mouse.

In the 32-bit option the mouse speed can be controlled in the Setup Parameters menu. To change the speed, select Setup from the main menu and then select Parameters. Enter a new value in the mouse speed field.

You can also move the cursor using the four cursor keys available on most enhanced keyboards. The cursor moves one user-defined grid step in the direction of the arrow each time an arrow key is selected.

THEORY OF OPERATION

This section explains how to use PADS-Logic to create your schematic drawings. Consider this a "road map" for creating a schematic.

Periodically as you work, you should save the results to a schematic file with the Job Out command.

1. First, define the drawing size you want to use. The default size is the US D format. To change this use the Parameters command in the SetUp menu. You can also define other standard defaults with this command. We suggest setting the working grid to 100 with the G Modeless command, and setting your standard color assignments with the SetUp command.
2. To create your schematic, begin by adding part types on sheet one, using the Add Part and Copy commands. Add part types by typing their names, or select them with the library browse command. You can rotate or mirror the part types before positioning them.
3. Part types are automatically assigned reference designator names and pin numbers, and multi-gate part types are assigned gate identifiers. These names may be changed with the Edit Text command. Use this command to assign values to part types like resistors and capacitors as well.
4. If you need a part type that is not in the library, use the Part Type command in the Hier/Lib menu to create it. Part types consist of two data types; electrical information and a CAE decal. CAE decals can be shared by other part types, so you won't have to duplicate them. When you enter the part type editor, your schematic is temporarily saved, and then reloaded when you exit from the part type editor.
5. Connections are made using the Add Conn command. Connections must start and finish on electrical connect points. Connections can go be-

tween part type or connector pins, to other connections, to off-page flags, busses, or to ground and power symbols. Connections are assigned default net names when they are added to the schematic. These can be changed with the Edit Text command. Connections between sheets may be made with off-page references. The net name is used to connect them together when the net-list is generated.

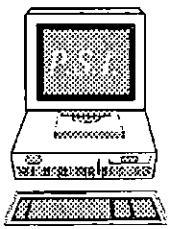
6. Busses may be added with the Add Bus command. Once they are created, connections can be made to them with the Add Conn command.
7. You can move part types, connections, and other items with the Move command. Buses are moved with the group command.
8. If you need to add an additional drawing sheet for a schematic, use the Add Sheet command in the In/Out menu. Since PADS-Logic uses a multi-sheet database, you do not have to save the first sheet. You can switch between sheet 1 and sheet 2 with the SH modeless command.
10. Groups of part types and their associated connections can be moved, copied, or deleted with the Group command. First define the group, then select the action to take on the group. If you wish to move a group of items across sheets, first use the Cut Group command, save the group, change to the target sheet, then paste the group.
11. Miscellaneous notes are added with the Add Text command.
12. When all parts and connections have been added, use the Net Status and Part Status commands to check the schematic. Potential errors are listed in these reports.
13. The Plot command creates an output file for a pen-plotter, matrix printer, laser printer, or photoplotter.

14. Create a netlist with the Net List command in the Reports menu. This net list can be read into PADS PCB design programs.

Technical Support

Technical Support will be happy to answer questions concerning PADS-Logic V2.5. The hotline number is 1-800-423-PADS. (Have support number ready). International callers please dial (508) 486-3328. PADS Software's other access numbers are:

Modem\BBS: ..... (508) 486-9549

Sales: 1-800-255-7814

FAX: ..... (508) 486-8217